The thesis of Daniel John Reyna is approved:

Phyllis K Russell

Dr. Sidney H. Schwartz

Dr. Timothy W. Fox, Chair

California State University, Northridge
TABLE OF CONTENTS

iii

LIST OF FIGURES...........................................................................................................ii
LIST OF TABLES...............................................................................................................iv
ABSTRACT.........................................................................................................................v
Chapter 1 Introduction ......................................................................................................1
  1.1 The need for Computational Fluid Dynamics .........................................................1
  1.2 Transonic Flow as an Instructional Tool .................................................................1
  1.3 Circular Cylinder ..................................................................................................3
    1.3.1 Flow Characteristics .......................................................................................3
  1.4 NACA 0012 Airfoil ...............................................................................................6
    1.4.1 Flow Characteristics .......................................................................................6
Chapter 2 Analysis ............................................................................................................8
  2.1 Circular Cylinder ..................................................................................................8
    2.1.1 Grid Considerations .......................................................................................8
    2.1.2 Determination of Convergence ......................................................................10
  2.2 NACA 0012 Airfoil ...............................................................................................15
    2.2.1 Grid Considerations .......................................................................................15
    2.2.2 Determination of Convergence ......................................................................19
Chapter 3 Results ..........................................................................................................21
  3.1 Circular Cylinder ..................................................................................................21
    3.1.1 Flow Coefficients .........................................................................................21
    3.1.2 Bow Shock Detachment Distance .................................................................30
    3.1.3 Normal Shock Relations ..............................................................................34
  3.2 NACA 0012 ..........................................................................................................36
    3.2.1 Flow Coefficients .........................................................................................36
    3.2.2 Grid Dependence and Adaptation .................................................................42
Chapter 4 Conclusion ....................................................................................................48
REFERENCES.................................................................................................................49
APPENDIX A Airfoil Coordinates .................................................................................52
APPENDIX B Fluent Transonic Airfoil Tutorial ............................................................55
APPENDIX C Fluent Transonic Cylinder Tutorial .........................................................73
LIST OF FIGURES

Figure 1: Supersonic Cylinder Shadowgram (From Ref. 7) ................................................. 4
Figure 2: Bow shock detachments distance (From Ref. 9) .................................................. 5
Figure 3: Airflow around an airfoil in transonic flow (Ref. 5) ............................................. 7
Figure 4: Cylinder domain shape and dimensions (not to scale) ........................................... 8
Figure 5: Structured grid example showing increasing cell sizes ........................................ 9
Figure 6: Cylinder grid using symmetrical boundary ........................................................... 9
Figure 7: Structured and unstructured grids ......................................................................... 10
Figure 8: Supersonic grid convergence and migrating bow shock ........................................ 14
Figure 9: NACA 0012 airfoil .................................................................................................. 15
Figure 10: Open and closed trailing edges ............................................................................. 16
Figure 11: Airfoil domain shape and dimensions ................................................................. 17
Figure 12: Different grid resolutions for the NACA 0012 airfoil .......................................... 18
Figure 13: Mach 0.9 velocity contours .................................................................................. 20
Figure 14: Mach 0.5 velocity contours .................................................................................. 22
Figure 15: Cell improvements outside of the boundary layer ............................................... 23
Figure 16: Drag Coefficients at all Mach numbers compared to experiment ....................... 24
Figure 17: Comparison of results to that of the literature and experiment ......................... 25
Figure 18: Comparison of pressure and skin friction coefficient ........................................ 29
Figure 19: Comparison of pressure and skin friction coefficient 2nd Order ......................... 30
Figure 20: Supersonic flowfields for the circular cylinder .................................................... 32
Figure 21: Supersonic bow shock detachment distance, experimental data from Ref. 2 ....... 33
Figure 22: Mach 1.3 density contours ................................................................................... 34
Figure 23: Comparison of computed normal shock relations to theory ............................... 35
Figure 24: Airfoil subsonic flowfields .................................................................................... 39
Figure 25: Subsonic airfoil pressure distributions with experiment ....................................... 41
Figure 26: Grid dependence of pressure coefficient ........................................................... 43
Figure 27: Vector and contour plots showing the shock and boundary layer ....................... 44
Figure 28: Comparison of refined grid to experiment ......................................................... 45
Figure 29: Grid changes as a results of refinement ............................................................... 46
LIST OF TABLES

Table 1: Cylinder grids properties.................................................................10
Table 2: Airfoil grid properties.......................................................................17
Table 3: Cylinder drag coefficients.................................................................21
Table 4: Airfoil calculated lift coefficients......................................................36
Table 5: Airfoil calculated drag coefficients....................................................42
ABSTRACT

TRANSONIC FLOW PROBLEMS
AS CFD INSTRUCTIONAL TOOLS

By

Daniel John Reyna

Master of Science in
Mechanical Engineering

The possibility of using compressible flow problems as an instructional tool for teaching techniques in computational fluids in engineering courses and laboratory environments was explored using a circular cylinder and a commonly studied NACA airfoil. Based on experience with problems and shortcomings of some teaching material used in past coursework, the cylinder and airfoil were examined from the perspective of a student of CFD. For each case, a mesh was created, a solution obtained, and both the process and results examined for agreement with experiment. From there, either the problem setup or the solution process was adjusted to improve the results. New generalized tutorials were written for the cylinder and airfoil problems that may be used in a classroom or laboratory setting for student assignments. In this material, a step-by-step process is outlined to guide the introductory CFD student in obtaining solutions using FLUENT and overcoming difficulties that may be encountered during this process. Key items include problem setup, solution stability, solution convergence and examination of results to determine if the results are valid based on previous coursework. Following the process, further study is encouraged through variation of the problem setup to examine fundamental aerodynamic and CFD principles.
Chapter 1  Introduction

1.1  The need for Computational Fluid Dynamics

Throughout the history of aviation, the need to reliably predict performance characteristics of an aircraft design was necessary. Will a given design be stable? Will it have enough thrust? Will it meet range specifications? For a majority of the past 100 years of aviation, wind tunnels were the only practical method of design test and verification. Wind tunnels provided a volume of controlled airflow in which to measure the aerodynamic coefficients of everything from wing sections to entire airframes. However, wind tunnels require large, expensive buildings with high operating costs. The development of practical computational fluid dynamics grew out of the need to analyze various design elements under different flow conditions without the associated costs of wind tunnels. Flow parameters like freestream velocity, angle of attack, Mach number, and Reynolds number can be varied along with static conditions like pressure, temperature, and density to simulate effects like altitude. All this can be done without the need to build models containing complex measurement equipment and installing them in wind tunnels for testing at high cost.

Combined with Computer Aided Drafting (CAD), it is relatively simple to go from a computer model to the computational domain using a number of commercial or custom software applications. This, however, is not to say that the days of wind tunnel testing are over. Wind tunnels still provide valuable information that cannot currently be replaced in its entirety. Validation of computational code and methods of analysis is crucial. Without validation, CFD results are essentially worthless for use in design work. What computational fluids does allow is for the engineer to analyze many different designs of a component or structure in a short period of time with a fraction of the cost. This helps determine whether or not to include a design aspect in the final product or test setup. Once an optimal shape is determined in the computational domain, this can then be verified in the wind tunnel and put into the final design.

CFD doesn’t exactly come without its own associated costs. Simple two-dimensional analyses don’t present too many problems with modern computational resources. Most, including the analysis done for this study, can be done on small desktop computers without much difficulty. Moving to larger and more complex three-dimensional problems that include sophisticated turbulence and heat transfer modeling can require days or weeks to prepare, set up, and execute on very large dedicated systems and clusters of computers. Regardless of the necessary tools, CFD represents an indispensable part of an engineer’s toolbox.

1.2  Transonic Flow as an Instructional Tool

Covering the variety of uses of CFD is not possible and outside the scope of this paper which involves aerospace applications. That being said, a common area in which computational fluids is playing a big role in design analysis is in transonic flows (Ref. 11). Compressible flow results in a variety of difficult problems affecting every major aspect
of engineering modern jet aircraft like airline and cargo transports that operate at high subsonic Mach numbers. Because of this modern-day impact, students must learn to recognize compressible flow problems and be able to understand the source of aerodynamic changes that occur when a moving body approaches the speed of sound and even after exceeding it. This study is intended to introduce basic CFD solution techniques to the reader and to explore the use of CFD as a supplement to traditional teaching techniques. This should provide the student familiarization with the CFD environment and some understanding of the capabilities and limitations of its use because computational fluid dynamics is not a trivial approach to engineering. A user cannot simply input the flow conditions, push enter, and expect the results to be accurate. A great deal of pre-processing work is necessary in order to tailor the flow grid to what conditions should be present in a real flow. Whether it’s solution Mach number, Reynolds number, angle of attack, or body geometry just to name a few examples. Even with the most diligent problem preparation, students of CFD must also be able to relate theory to what is presented to them once a solution has been found as there is no point in using erroneous results as a learning tool as well as an engineering one. The user must be able to look at a set of results and decide if they are valid and represent reality. If they do not, why? If so, are they the optimal results? For example, has the solution converged? One case encountered in this study demonstrates the latter situation in which convergence isn’t trivial and the solution appears valid, however, the solver must be allowed to continue further. The student must also experience the necessary methods of testing the solution for various causes of error, some of which are covered here.

For addressing multiple aspects of simulating transonic/supersonic flow, both blunt and streamlined objects were chosen. Both types of objects have characteristics that test the abilities of flow models in use today and provide the user with various challenges in obtaining solutions. While this isn’t totally relevant to the student of compressible flow, the experience with CFD limitations is very relevant. Most importantly, they show many interesting flow characteristics such as significant flow separation with strong recirculation in the wake region immediately behind it where the use of Euler-based models or approximations won’t be able to properly predict the separation point and therefore the pressure drag. Supersonic velocities will present strong shocks along with associated flow separation. Streamlined objects allow smooth flow across their length but with the addition of compressibility will test how well CFD can predict its effects. The presence of shocks also gives students the opportunity to explore grid dependence in solutions. The airfoil is expected to also display the interaction of shocks with the boundary layer that should be grid dependent. Blunt bodies will also display similar types of shocks as a streamlined body, however, the strong curvature of a blunt body makes shock boundary layer interaction difficult to examine.

A number of different methods and models exist for solving various flowfields. For this study, the CFD code Fluent by ANSYS was used. Fluent is a well-known tool representative of what is widely used in both industry and academia. Fluent provides the user with a choice of viscous models. These include Inviscid and Laminar, and a choice of the turbulence models including Spalart-Allmaras, k-ε, k-ω, and Reynolds Stress.
These models are fairly standardized among the majority of commercial codes. Additional models exist; however, many are specialized or not prominent enough to have made it into a major code like Fluent (Ref. 25). This is not to mean that this study is a test of any specific code or software package as much as it is a test of techniques for teaching CFD. Comparison of results to experiment is done as a means of testing methods used to obtain those results. For the CFD calculations to be accepted as valid for this analysis, specific characteristics of transonic and supersonic flow like bow shocks and surface shocks with associated flow separation should be present. Other quantifiable effects should be predictable from established compressible flow theory.

1.3 Circular Cylinder

1.3.1 Flow Characteristics

The classic example used in every fluid mechanics textbook, the circular cylinder in crossflow, has been extensively studied over the past century in every flow condition imaginable. These range from low Reynolds numbers to high Reynolds numbers, and subsonic to supersonic flows. This extensive amount of experimental data provides a good reference with which a student can validate CFD solutions. The circular cylinder flow field is also quite complex at all Reynolds numbers and Mach numbers.

For the simulations presented here, the freestream Mach number was varied in order to observe the changes in the flow field characteristics. At the low end of the scale is flow at Mach 0.3 where the flow can be considered incompressible. This implies that the calculated drag coefficient should be verifiable with those experimental results at a similar Reynolds Number. The next is Mach 0.5 where the flow is no longer incompressible and localized flow velocities could reach Mach 1.0. Subsonic velocities conclude with Mach numbers of 0.75 and 0.9. Next is sonic ($M = 1.0$) and then supersonic, $M = 1.15$, 1.3 and 1.546. The odd value of 1.546 was considered in order to directly compare results to investigations by Alperin (Ref. 2).

At low Mach numbers, the circular cylinder should have highly accelerated flow around the top surface with separation immediately aft. A significant wake downstream of the trailing surface should also display large areas of circulation and possible oscillatory shedding, (Ref. 27). This wake is an important feature when creating the grid and setting up the solver. Higher Mach numbers bring increased velocities around the circumference until at approximately Mach 0.5 the flow reaches sonic velocity at the top of the cylinder. Above this recompression shocks should be present with associated flow separation. This is the first appearance of shocks in this study. From data on airfoils, the location of the shocks should move further aft reducing the width of the wake (Ref. 4).

Freestream Mach numbers exceeding unity will produce a bow shock upstream of the leading edge, (Ref. 5). The large blunt radius produces a broad shock that is detached from the surface with a hyperbolic shape. If the flow accelerates back to supersonic velocities on the surface, there should exist additional normal surface and downstream oblique shocks from recompression along the cylinder surface. Figure 1 shows a
shadowgraph of supersonic flow around a cylinder showing the bow shock, recompression shock, flow separation, and two trailing shocks on either side of the wake. This bow shock moves closer in towards the cylinder as the Mach number increases. Figure 2 illustrates the relationship between Mach number and detachment distance. The exponential decay of the distance implies a bow shock infinitely far from the cylinder. Computational simulations at Mach one should not produce a visible bow shock.

The appearance of the flowfield in Figure 1 shows a “dead air region” immediately behind the body of the cylinder. This dead air is a location of separated flow and recirculation. A student can examine flow vectors along the circumference to see the boundary layer changes that lead to flow separation along both the top and bottom of the cylinder with the arrows reversing direction. This expectation of separated flow should be an indication to the user that concentrated grid meshing should be placed there to properly resolve the circulation.

Figure 1: Supersonic Cylinder Shadowgram (From Ref. 7).
Figure 2: Bow shock detachments distance (From Ref. 9)

Flow across the bow shock will be decelerated to subsonic velocities and follow the normal shock relations, (Ref. 9):

\[ M_2^2 = \frac{1 + \left(\frac{\gamma - 1}{2}\right) M_1^2}{\gamma M_1^2 - \left(\frac{\gamma - 1}{2}\right)} \]

\[ \frac{P_2}{P_1} = 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1) \]

\[ \frac{T_2}{T_1} = \left[ 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1) \right] \frac{2 + (\gamma - 1) M_1^2}{(\gamma + 1) M_1^2} \]

\[ \frac{\rho_2}{\rho_1} = \frac{M_1^2}{1 + \frac{\gamma - 1}{\gamma + 1} (M_1^2 - 1)} \]

where the subscripts 1 and 2 represent the conditions upstream and downstream of the shock, respectively. It can be seen that the flow properties are simply a variable of the freestream Mach number.
1.4 NACA 0012 Airfoil

1.4.1 Flow Characteristics

Simulation of a streamlined body like an airfoil provides the CFD student with a number of opportunities to explore CFD and transonic flow simulation without the difficulties associated with the strongly curved cylinder. Like the cylinder in cross flow, shock development and migration will be present, however, the nature of the shock behavior can vary depending on a variety of factors like geometry, freestream velocity and angle of attack. Given the small amount of surface curvature along the upper and lower surfaces, boundary layer changes and flow interaction with normal shocks are observable. These strong gradients and variations in flow properties will produce grid dependence that may necessitate a student to look into grid refinement.

The NACA 0012 is one of the most thoroughly studied individual airfoil shapes, (Ref. 15). Considered the standard for evaluation, there exists an enormous amount of information available in nearly all flight regimes, Reynolds numbers, and flow conditions making it a perfect choice for comparison of CFD with experiment and for investigation by a student studying CFD. The result is that much of this information can be used today for testing, and for code validation. Being an early four-digit NACA shape, the 0012 airfoil is a symmetrical airfoil with zero camber, twelve percent thickness ratio, and maximum thickness occurring at thirty percent chord. The angle of attack for the airfoil simulations is set at a constant two degrees, positive.

The use of incidence angle with the flow, as opposed to no angle of attack, was done to examine the differences in flow characteristics between the upper and lower surfaces as the Mach number increases. As a result of the increased flow velocity on the upper surface, the critical Mach number will be reached sooner than for the lower surface (Ref. 5). It is possible to observe a similar phenomenon with cambered airfoils, with or without angle of attack. Simulations under these flow conditions will also allow the student to compare their CFD simulations with experimental data as the freestream Mach number exceeds the critical value, $M_{cr}$, and begins to approach sonic velocity. Figure 3 shows the expected changes in flow around an airfoil with increasing Mach number.

![Diagram showing flow characteristics at M = 0.75 and M = 0.81](image_url)
Figure 3: Airflow around an airfoil in transonic flow (Ref. 5).

Below the critical Mach number, the flow is subsonic everywhere. Lift and drag characteristics (Ref. 1) are expected to be typical of those with similar Reynolds numbers. As the freestream reaches critical velocity, the flow over the top surface reaches the speed of sound along with an associated increase in drag. When the local flow exceeds Mach one, a recompression shock forms. This will be seen as a sharp discontinuity in the pressure coefficient and significant flow separation. Increasing Mach number will eventually lead to formation of a lower surface shock and push the shock on the upper surface towards the trailing edge, (Ref. 5). The lower surface flow will terminate in a shock at the trailing edge. The decrease in lift is due to the shallower lift-curve slope caused by flow separation aft of the normal shock on the upper surface combined with the relatively smooth flow along the lower surface, (Ref. 5). The flow separation increases the overall pressure on the upper surface compared to that of the lower surface with its relatively smooth flow. The drag is expected to increase exponentially towards Mach one. These surface shocks, as seen in the results, provide an excellent opportunity for a student of CFD to see the effects of grid resolution and localized grid adaptation. Supersonic freestream velocities also lead to the formation of a bow shock similar to that seen upstream of the cylinder in Figure 1; however those cases are skipped in this study since much of the important supersonic effects are seen in the cylinder cases.
Chapter 2  Analysis

2.1  Circular Cylinder

2.1.1  Grid Considerations

For any CFD simulation, the user must tailor the grid to the geometry and the flow characteristics expected. This includes the overall shape, the size, and resolution. The resolution isn’t about how fine the grid is everywhere as much as it is about where the cell size is tailored to match the flow. The CFD user should recognize that a very fine grid can require large amounts of computational resources. If the same results can be obtained using a fraction of the computing power then there may be no reason to spend the extra time with the grid.

The first important aspect that deserves attention is in the domain. While the domain shape may or may not have much affect on the results, the ability to easily tailor the grid to the flow should be considered. The simplest domain shape is an O-type grid with an overall circular shape that extends far enough to allow proper development of the flowfield. This is the easiest to create and with radial lines little problems will arise in cell shapes. However, depending on how the user applies grid points along the surface, this type of grid will have uniform cell distribution moving radially outward. The characteristics of the flow downstream need concentrated cells downstream more than upstream of the cylinder. Excluding significant portions of the affected flow will create large errors in the solutions. The same grid will also be used for supersonic flow conditions so a large enough domain upstream needs to be created to allow for the expected bow shocks upstream of the surface to be viewed. Figure 4 has the domain shape and dimensions chosen.

The portion of the grid from the center of the cylinder forward resembles a simple O-grid with cell size migrating radially outward. The rectangular region aft of the center made the grid relatively easy to produce by allowing consistent, easily meshed boundaries.

Figure 4: Cylinder domain shape and dimensions (not to scale).
Close to the cylinder wall, the grid cells must be concentrated in order to properly resolve the boundary layer and other small-scale flow effects. With a cylinder diameter of one meter and a uniform cell distribution, the first cell extends out from the wall one millimeter. Cell growth in the boundary layer is 10% in the boundary layer extending seven rows out. Expected behind the cylinder is significant wake activity extending from the cylinder all the way to the downstream boundary. Cell size is concentrated closer to the midline of the domain with cell growth both downstream and off axis to reduce cell count.

![Structured grid example showing increasing cell sizes.](image)

Figure 5: Structured grid example showing increasing cell sizes.

In total, four separate grids were created for testing solution grid dependence. The first three grids contain the common characteristics mentioned above. Differences in the grids were created in order to examine the solutions for grid dependence and for a qualitative look at solution stability. The first is a structured grid that takes advantage of the symmetry involved with a circular cylinder and its geometry. In many cases, using symmetrical boundary conditions allows the computational domain to be reduced by a factor of two. This significantly reduces the computational time and resources necessary to compute the solution. Also, the symmetrical grid was modified to extend further out in terms of the cylinder diameter to explore a feature found in solutions with the full-sized grids (see Chapter 3.1.2).

![Cylinder grid using symmetrical boundary.](image)

Figure 6: Cylinder grid using symmetrical boundary.
The second grid is also structured, this time with a complete domain and no symmetry involved. Cell concentration near the surface of the cylinder was increased in order to more completely resolve the boundary layer. The third is a hybrid grid with quadrilateral cells making up the boundary layer and tetrahedral cells the balance of the domain. A hybrid mesh was chosen as opposed to a purely unstructured grid which is unable to completely resolve the boundary layer and still maintain reasonably low node counts, (Ref. 27). Basic grid characteristics are listed in Table 1 below. The fourth grid in the table was found to be necessary after comparison of initial results to experiment. It was created with a much finer cell distribution close to the cylinder however the boundary layer and the regions near the farfield boundaries were left relatively untouched in order avoid an overly excessive cell count. Issues were encountered early on with larger symmetrical grids and unstable solutions. Converged solutions were not able to be obtained for comparison with equivalent full grids.

![Structured and unstructured grids.](image)

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Type</th>
<th>Cell Count</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>symmetrical</td>
<td>Structured</td>
<td>34641</td>
<td>28</td>
</tr>
<tr>
<td>coarse</td>
<td>Structured</td>
<td>85000</td>
<td>28</td>
</tr>
<tr>
<td>tetrahedral</td>
<td>Hybrid</td>
<td>52519</td>
<td>27</td>
</tr>
<tr>
<td>fine</td>
<td>Structured</td>
<td>101000</td>
<td>27</td>
</tr>
</tbody>
</table>

Table 1: Cylinder grids properties.

2.1.2 Determination of Convergence

A significant component of computational fluid dynamics is the production of a properly converged solution. A large amount of work put into producing a quality grid can be at a loss if a properly converged solution cannot be obtained. Several strategies exist to control the solution to help aid in convergence. Many of these are different ways to reduce the speed at which the flowfield develops in the domain to a level that the solver can handle. They include turning off equations (energy, momentum, turbulence), reducing under relaxation factors, and even starting the final solution using a partially solved one. Each of these methods was used at least once to solve for the cylinder flowfield. Once a stable solution is proceeding, determining convergence to the final
solution can be deceiving. The two important methods used in this study are residual and force monitoring. In residual monitoring, the user allows the solution residuals to reduce several orders of magnitude. When the flowfield converges, it becomes fixed and steady, implying that all flow-induced force values like lift and drag will become constant.

For solution convergence, residual monitoring was found to be somewhat unreliable. Because of the geometry and flow characteristics encountered, heavy use of under-relaxation factors became important and lead to widely variant residual plots. Each change in the under-relaxation factors resulted in a 1-2 magnitude increase in the residuals followed by a gradual decay. Monitoring of the most dominant force coefficient, drag, proved to be the most reliable method of determining convergence. When reducing the values of under-relaxation factors, convergence can become slowed causing the force coefficients to change more slowly. A student or inexperienced user may mistake this slow movement for a converged solution. For all solutions presented here, convergence was determined to have occurred when no significant change in the drag coefficient was seen for at least 20 iterations, but only with under-relaxation factors at the default values for Fluent. This isn’t to say that the residuals were completely ignored. Even with a converged drag coefficient, certain residuals had to remain low for the solution to have been considered converged. One case shown later on is one instance where converged force values and high residuals were encountered along with the resulting solution discontinuities.

Figure 8 shows a supersonic flowfield around the cylinder at six different points in the solution. Initially, the shock, shown as a strongly increasing density gradient upstream of the cylinder, is close. As the solution progresses, the shock moves further upstream and major wake characteristics develop. Eventually the detachment distance stabilizes. The final two in the sequence have similar downstream regions but bow shocks at different distances. This implies that the wake has reached convergence sooner and that CFD users can observe bow shock detachment distance to indicate convergence when simulating supersonic flows around blunt objects.
a) 1 Iteration

b) 9 Iterations
c) 17 Iterations

d) 29 Iterations
Figure 8: Supersonic grid convergence and migrating bow shock.
2.2  NACA 0012 Airfoil

2.2.1  Grid Considerations

As mentioned before, significant effort must be made to create a computational grid that properly resolves and represents the flow conditions that are present. Creation of a computational grid for an airfoil shape includes many similarities to that of the cylinder. There must be considerations for the boundary layer, farfield, and wake regions and for the expected flow effects. Unlike the cylinder, the airfoil has a surface curvature that varies with distance upstream of the leading edge. Figure 9 is a diagram of the NACA 0012 airfoil shape. The leading edge has a much smaller radius of curvature than the cylinder with a relatively flat region after the point of greatest thickness, ~0.30c. The CFD student should already be aware that areas with more curvature will show more variation in the flow. The trailing edge is a sharp point that should lead to specific compressibility effects at high transonic freestream velocities, (Ref. 5), mentioned in Section 1.4.1. It is this knowledge and expectations of the flowfield that allow the CFD user to tailor a grid to the flow and concentrate cells around notable areas where specific flow changes are expected.

![Figure 9: NACA 0012 airfoil.](Image)

A MATLAB function, Appendix II, was used to calculate the correct x-y coordinates necessary for mesh generation of the airfoil. The formula used for the NACA geometry is, (Ref. 1):

\[
\pm y = \frac{t}{0.20} \left(0.29690 \sqrt{x} - 0.12600 x - 0.35160 x^2 + 0.28430 x^3 - 0.10150 x^4\right)
\]

This equation for a symmetrical NACA airfoil produced a shape with an open trailing edge, (Figure 10, left). While this would no doubt represent a more realistic shape on an aircraft with a finite edge radius, the equation was modified to simplify pre-processing and grid creation. To do this, the final term in parenthesis was changed from 0.10150 to 0.10360. Figure 10 shows the difference in resulting shapes. Appendix II includes both the MATLAB function and grid points used for creation of the airfoil.
The upstream boundary is an O-type placed a distance of fourteen chordlengths from the leading edge. The wake region extends a distance of 26c downstream. To account for the difference in curvature over the surface of the airfoil, a bell-shaped scheme was used to concentrate cells near the leading and trailing edges. This was deemed necessary because of the strength of the expected gradients and flow changes in those locations that must be resolved properly. It also prevents excessively skewed grid cells by keeping radial grid lines normal to the farfield boundaries. Skewness increases the uncertainty in locating the center of each grid cell, decreasing the solution’s accuracy and stability, (Ref. 27). More advanced grid generation software is able to automatically vary the cell boundaries such that cell skewness is kept at a minimum.

The trailing surfaces have much less curvature than the leading edge of the airfoil and much less than the cylinder. Yet, strongly turbulent boundary layer and flow separation is expected to be present in these locations as a result of shock formation on the upper surface. The location and shape of surface shocks is not known beforehand so increasing cell concentrations around the shock location is not possible. Instead, grid refinement was explored during the solution process as a means of increasing accuracy without stressing already finite computational resources. In Fluent, this can be accomplished through different methods. The refinement can be done in specific regions, for example, around the surface boundary extending outward as far as the user desires. Another method is to use a lower resolution solution as the basis for locating areas to concentrate refinement.
A line of concentrated cells was created extending from the trailing edge to the downstream boundary to resolve the wake. Cell resolution decreases further above and below the chord axis as well as downstream of the trailing edge to reduce cell count away from locations where larger flowfield gradients are expected. The line of trailing cells was made parallel to the chordline since this study only considers small angles of attack. If the angle of attack is expected to place the expected wake well away from the chordline, the user should angle the wake mesh to compensate. Simply rotating the wake region up or down may not be ideal. In extreme cases, the farfield boundaries should also be altered in order to avoid overly skewed cells.

As for the cylinder, different airfoil grids were created to examine solution grid dependence. Any strong grid dependence should show its presence around the shocks expected at higher transonic Mach numbers, (Ref. 23). The grid scaling was based on the number of cells on the surface of the airfoil. The coarse grid places 75 cell faces on the upper and lower surfaces. A medium grid nearly doubles the cell count by decreasing the distance between mesh points by 33%. Finally, a fine grid places a total of 300 cell faces on the airfoil surface. Cell thickness in the boundary layer was also manually refined during the solutions to explore grid dependence in resolution of the boundary layer. Table 2 shows grid properties.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Type</th>
<th>Surface Nodes</th>
<th>Cell Count</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>Structured</td>
<td>75</td>
<td>22800</td>
<td>55</td>
</tr>
<tr>
<td>Medium</td>
<td>Structured</td>
<td>100</td>
<td>40200</td>
<td>44</td>
</tr>
<tr>
<td>Fine</td>
<td>Structured</td>
<td>300</td>
<td>120000</td>
<td>17</td>
</tr>
</tbody>
</table>

Table 2: Airfoil grid properties.
Figure 12 illustrates the differences in the grids used for the simulation of flow around the airfoil. Different characteristics of the grid are visible aside from the cell sizes. Common to all of the grids is the decreasing cell size close to the surface as in the cylinder grids. Unlike the cylinder, the airfoil does not have a constant radius of curvature around its perimeter. Areas with more curvature like the leading edge are expected to create notable flow changes. The grid resolution was increased around the leading edge in order to accommodate these expected variations. In the second grid, the transition from the boundary layer to the cells in the rest of the domain is not even nor is the distribution of cells upstream of the leading edge. This grid was discarded in favor of the third grid above, Figure 12 c). The cell distribution around the area of the leading edge has been smoothed out to produce the more ideal cell distribution while keeping the cell count the same. Unfortunately this also induced more cell skewness. The radial cell boundaries do not intersect the airfoil surface perpendicular to it. For simulation purposes this skewness was determined to be acceptable since the leading edge is expected to be an area with the least interesting flowfield. The fourth grid is the highest resolution used.

For qualitative examination of shock formation and other flow characteristic, the coarser grids were used in order to save computational resources. This allows significantly reduced time to obtain a solution. From the previous results with the cylinder, no significant reduction in accuracy was expected in this respect.
2.2.2 Determination of Convergence

Determination of convergence for the airfoil solutions was very similar to the cylinder cases, by monitoring residuals and force coefficients. The NACA 0012 airfoil is symmetrical in its shape just like the cylinder, however, because it is elongated it is possible for it to have an angle of attack with respect to the flow. This angle of attack gives the body lift. For the cylinder, a plot of drag was monitored during the solution until it had stabilized at or very near a single value. The degree of convergence of lift and drag did not occur at the same rate implying that it was necessary to monitor more than one force value during the solution.

Unlike the circular cylinder, an airfoil is a streamlined object implying steady flow with little to no significant turbulence or flow separation. Convergence for the circular cylinder required significant reduction of under relaxation factors to prevent the solution from proceeding at a rate too quick for Fluent to adapt leading to divergence of the solution. The airfoil, being streamlined, required no reduction in under relaxation factors and could be completed with many fewer iterations. Beginning a solution using a first-order discretization and switching to second order when the solution was very near convergence also improved solution stability and accuracy. As with any changes in the FLUENT solution controls, this resulted in an immediate increase in residuals and jumps in the monitored force coefficients. These values diminished to the levels they were prior to the changes.

This isn’t to say that all was without difficulty. The highest velocity examined for the airfoil, \( M = 0.9 \), required a significant amount of coaxing for a stable solution to be found. This in turn required more iterations than any even of the cylinder solutions. It is unknown whether switching over to a second-order scheme improved accuracy as it made the solution unstable and divergent. No second-order solution was obtained. Interestingly, the solutions for \( M = 0.75 \) converged quickly with little effort even with the presence of the strong shock on the upper surface and resulting flow separation. It was apparent that the solver was unable to handle the presence of more than one strong shock as well as the larger separation and wake regions. Something that arose in the literature was, at least for the cylinder, all evidence of periodic shedding and vortex in simulation activity disappears by Mach 0.9, (Refs. 6 and 17). If this translates over to a streamlined body like an airfoil, this seems more likely to make the solution stable since the flow would be expected to be less time dependent. This gives more credence to the shock explanation. Further exploration of obtaining this solution is left open for future study.

Attempts to solve the Mach 0.9 flow using the fine grid were completed with little success. The solution continuity residuals remained high, on the order of one, even with the flow coefficients converging on a single value. The velocity contours were examined for any adverse effects of the high continuity residual. The upper shock shows what appears to be rippling and distortion in the flowfield and the shock is located slightly further downstream (Figure 13). It is easy for a Fluent user to mistakenly recognize the convergence of the lift and drag values as a successful solution. However, the discontinuities in the flowfield and high continuity residuals point to an obvious problem
with the solution. For the fine grid results to be considered valid, all efforts should be made to reduce the residuals for a smooth flow and to examine the flowfield for problems that point to an invalid solution.

![Figure 13: Mach 0.9 velocity contours showing effects of high continuity residuals.](image)

Figure 13 shows contours of velocity for two separate Mach 0.9 solutions, one in which the force coefficients had converged, but the residuals were excessive. The need for the CFD user to fully examine results for consistency is extremely important.
Chapter 3  Results

3.1 Circular Cylinder

3.1.1 Flow Coefficients

As mentioned in Section 2.2.2, the drag coefficient is the most important force coefficient for flow around a cylinder as lift is zero with the absence of circulation. By monitoring a plot of the drag coefficient throughout the iteration process, convergence was determined when the solution became constant to four significant figures. Table 3 shows drag coefficients for each of the three initial grid solutions from Mach 0.3 to 1.546. Note, the fourth grid is not included here as it was created specifically to examine some of the results show below.

<table>
<thead>
<tr>
<th>( M )</th>
<th>Symmetrical</th>
<th>Full Grid (coarse)</th>
<th>Full Grid (tet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.2161</td>
<td>0.2947</td>
<td>0.2927</td>
</tr>
<tr>
<td>0.5</td>
<td>0.4302</td>
<td>0.4674</td>
<td>No solution</td>
</tr>
<tr>
<td>0.75</td>
<td>1.0981</td>
<td>1.1093</td>
<td>1.0998</td>
</tr>
<tr>
<td>0.9</td>
<td>1.5614</td>
<td>1.4845</td>
<td>1.6865</td>
</tr>
<tr>
<td>1</td>
<td>1.7553</td>
<td>1.7859</td>
<td>1.7821</td>
</tr>
<tr>
<td>1.15</td>
<td>1.6112</td>
<td>1.643</td>
<td>1.6255</td>
</tr>
<tr>
<td>1.3</td>
<td>1.5303</td>
<td>1.5485</td>
<td>1.5589</td>
</tr>
<tr>
<td>1.546</td>
<td>1.4787</td>
<td>1.4853</td>
<td>1.4813</td>
</tr>
</tbody>
</table>

Table 3: Cylinder drag coefficients.

All three grid resolutions show relatively good agreement through most of the range of Mach numbers with the symmetrical grid having slightly lower drag values at all Mach numbers. The exceptions are at \( M = 0.3 \) and 0.9. A proper solution using the unstructured grid was not found for \( M = 0.5 \). In fact, all solutions at this Mach number showed very high instability. The use of under-relaxation factors to control the solution helped, however, convergence was extremely slow even for those cases that did converge. The user must be able to recognize this slow solution advancement, as it is easily possible to mistake the slow movement of monitored values like drag and lift for solution convergence.

Examining the flowfield for the \( M = 0.5 \) case shows a possible reason for the difficulty in obtaining a solution. The flow around a cylinder accelerates as it moves around the circumference of the body up until the point where the surface is perpendicular to the farfield flow direction. Examining a contour plot of the velocity around the cylinder reveals that the flow reaches approximately 366 m/s. This is just above the speed of sound at standard conditions and may have contributed to the solution instability.
Figure 14: Mach 0.5 velocity contours. Maximum velocity 366 m/s.

The difference between the half grid and the other two at $M = 0.3$ may be a result of the lower overall resolution. At $M = 0.9$, no agreement between the three grids exists. Close examination of the three grids showed poor meshing in the areas immediately outside of the boundary layer region. The sizes of grid cells adjacent to the boundary layer region increased nearly 400%, too great for proper resolution of the flow. A fourth grid with a total of 101,000 nodes was created to validate the hypothesis that grid resolution immediately outside of the boundary layer was responsible for discrepancies in the results. Cell size immediately outside of the six-cell deep boundary layer meshing transitions smoothly. The boundary layer regions of both the full structured grid, a), and the improved one, b), are shown below in Figure 15.

The difference between these two grids can be seen in the smooth change in cell size moving radially out from the cylinder surface. For the two Mach numbers in dispute, $M = 0.3$ and 0.9, the drag coefficients were calculated using the finest grid to be 0.3766 and 1.482, respectively. Comparison of the $M = 0.3$ solution with the Delayed Detached-Eddy Simulations of Squires, et. al. at nearly identical Reynolds Number shows excellent agreement. The DDES drag coefficient for their fine grid simulation is 0.38, (Ref. 26). The resulting drag coefficient for $M = 0.9$ agrees remarkably well with that of the coarser full structured grid however no experimental value could be found that it could be compared to. For the upper portion of the study at $M = 1.546$ the solution value of 1.48 is quite close to the experimental value of approximately 1.45 obtained by Alperin in the wind tunnel, Ref. 2. In fact, as Figure 16 shows, CFD results for the drag coefficient are very close to those of experiment in the supersonic flows.
Figure 15: Cell improvements outside of the boundary layer.
Figure 16: Drag Coefficients at all Mach numbers compared to experiment. Experimental data taken from Ref. 6 and Ref. 17.

The obvious divergence from the subsonic experimental values is unexplained at this time. The appearance of the experimental results above mirror those found by Gowen and Perkins (Ref. 7) at higher subsonic Mach numbers. Interestingly the lower fineness ratio rocket tests referenced by Gowen and Perkins have a smoother subsonic trend that decays in a manner similar in shape, though not as steeply, as the Mach number decreases. Results plotted in Figure 16 are approximately 10% greater at Mach one than experiment but much closer to experiment at all supersonic Mach numbers. Why the code so greatly underpredicted drag in subsonic flows but is extremely close to supersonic experiments is most likely a result of the manner in which FLUENT solves for these two cases. The complexity of the subsonic elliptical scheme used for is significantly greater than that of the explicit marching technique used for supersonic flows. Since this study isn’t meant as a validation of CFD codes, it was found that a comparison of the results to CFD simulations made by others might be more relevant.

Figure 17 below left shows plots of surface pressure coefficient along with those of Squires earlier experimental measurements made by Roshko and van Nunen. The fine grid gives good agreement with the efforts of Squires, et al at similar Reynolds numbers. The results diverge slightly at the point corresponding to the location of highest velocity and downstream of the separation point. The point of separation is at nearly the same point around the circumference as in Ref. 26.
Figure 17: Comparison of results (blue) to that of the literature and experiment (Ref. 26). Top, pressure coefficient, lower, skin friction coefficient.
An interesting observation was made about the peak at the leftmost portion of the pressure coefficient plot. This point corresponds to the stagnation point of the cylinder, and exceeds the maximum value of unity allowed for incompressible flows as well as data from others. This implies that the assumption of an incompressible flow is incorrect and may be the cause of some observed discrepancies compared to the literature. A view of the peak pressure using a contour plot of pressure coefficient gives a value of 1.04 for \( Re = 1.7 \times 10^7 \) and 1.03 for \( Re = 3.6 \times 10^6 \). The major difference between the two Reynolds numbers is for the freestream Mach number. Verification of the pressure coefficient comes from compressible flow theory.

\[
\frac{P_0}{P} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\frac{1}{\gamma - 1}}.
\]

Where \( P_0 \) is the stagnation pressure, \( M \) is the Mach number and \( \gamma \) is the ratio of the specific heats. Rearranging this with the definition of dynamic pressure, \( P_0 - P \) or,

\[
Q = P \left( \frac{P_0}{P} - 1 \right) = P \left[ \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\frac{1}{\gamma - 1}} - 1 \right]
\]

A binomial expansion of the pressure relation gives a compressible dynamic pressure equal to,

\[
Q = \frac{1}{2} \rho v^2 \left( 1 + \frac{1}{4} M^2 + \frac{1}{24} \gamma M^4 + \ldots \right)
\]

The final term is the correction factor for determining the ratio of compressible dynamic pressure to that of incompressible flow. The pressure coefficient cancels out the incompressible dynamic pressure term leaving just the correction factor. With \( \gamma = 1.4 \), the pressure coefficient becomes,

\[
C_p = 1 + \frac{1}{4} M^2 + \frac{1}{40} M^4 + \ldots
\]

Additional terms aren’t significant and do not affect the calculation. The stagnation pressure coefficient for \( M = 0.3 \) is then calculated to be 1.02. This is somewhat less than that for the CFD solution result of 1.04. A Prandtl-Glauert transformation, where,

\[
\frac{C_p}{C_{p0}} = \frac{1}{\sqrt{1 - M^2}}
\]

gives a value of 1.05, better still. For \( M = 0.9 \), the CFD result for the stagnation pressure coefficient is about 1.24 with theory placing it at 1.22. Future CFD calculations with density constant may provide results in closer to agreement.
Figure 17, right, is a comparison of results for the skin friction coefficient with experimental measurements by Achenbach. The major peak in the skin friction is also associated with the calculated location of peak velocity. The sharp change in the shape of the calculated skin friction between 100 and 120 degrees corresponds to the point of separation. The results here are confirmed by the earlier calculations of drag, pressure, and skin friction coefficients by Squires, et al. Also, the prediction of the detachment location at 114 degrees by the higher quality grid is also in excellent agreement with the DDES results along the majority of the circumference of the cylinder (Squires, et al, 2008). The lower-resolution structured grid shows a detachment point further downstream at 118 degrees. Both the coarse and fine grids show little agreement with either the DDES calculations or experiment after the point of separation, a possible indication of poor resolution of the wake turbulence (Ref. 26). Squires, et al, did not address the poor agreement with experimental data for the skin friction coefficient. A student may desire to refine the grid locally to investigate this further.

Revisiting the difference in the drag coefficient for the different grids shows significant grid dependence for Mach 0.3. Figure 18 illustrates the differences in the results for the pressure and skin friction coefficients displaying the amount of grid dependence. The comparison to the results from Squires, et al was done using the finest grid resolution, however, the three other results deviate from those by more than a trivial amount. The plots of pressure coefficient show that the coarse and tetrahedral grids resemble each other closely, as was see in the close match of the drag coefficients. Interesting enough is that this case, $M = 0.3$, is the one with the least interesting flowfield characteristics. Since viscous effects dominate drag, the reader should initially expect solutions at this Mach number to display the least amount of deviation and grid dependence.

Fluent’s solution controls include higher order solution schemes to increase accuracy in simulations. The coarse and fine grid solutions were switched to second order discretization leading to convergence upon a drag coefficient of 0.367 (coarse) and 0.387 (fine), a near match to each other and other CFD solutions (Ref. 26). Figure 19 shows the surface pressure and skin friction coefficient results for the coarse and fine grids with for comparison. For pressure distribution grid dependence essentially disappears when a higher order solution scheme is used. The surface pressure coefficients are nearly identical. The only notable difference lies in the separation point with a slight dip in the coarse grid solution. The plot of skin friction appears significantly different around the location of greatest surface velocity. One can only suspect that this is due to differences in boundary layer resolution and especially the distance of the first nodal point from the cylinder surface; $y^+$. Interestingly, the fine grid was created to correct for the poor cell distribution immediately outside of the boundary layer mesh (Figure 15) not to improve the resolution inside of it. Mesh points around the circumference are identical and the overall boundary layer remained unchanged until 5 cells out radially from the surface. Major differences are only found out side of that. The resolution of this problem is left for later study. Suggestions include increasing the radial distance and number of cells in the boundary
layer meshing one or two points then obtaining a solution. Iterating this will find the point at which the flow outside of the boundary layer affects the skin friction within it.

Unfortunately, the agreement in force coefficients mentioned above calls into question results for the other Mach numbers since those were completed using first order. Additional iterations using second-order schemes were first tried on those Mach numbers with the greatest grid dependence, \( M = 0.3, 0.5, \) and \( 0.9 \). In each case the solutions converged on drag coefficients that were much more consistent with each other. Little grid dependence in force values was found once this was implemented. Before a user concludes that any degree of grid dependence is present, they must first ensure that the solutions must be completed using the most accurate methods available. Comparison to experiment is very useful in determining grid dependence and/or accuracy. Unfortunately, many computational problems can involve geometry for which no comparable experimental data exists.
Figure 18: Comparison of pressure (top) and skin friction (bottom) coefficient results for the different cylinder grids.
3.1.2 Bow Shock Detachment Distance

One cannot use CFD to explore transonic flow without including shocks. The flow around the cylinder at all Mach numbers showed no significant surface shocks that could be examined. The blunt shape produced easily distinguishable bow shocks in the solutions. The bow shocks were very characteristic to what is seen in Figure 1 and showed behavior expected from theory and experiment in which the bow shock distance decreased with increasing Mach number.

Comparisons of the converged supersonic solutions gave good agreement between numerical simulations done in this study and experiment. Figure 20 shows the flow field for $M = 1.0, 1.15, 1.3,$ and $1.546$ showing the decrease in detachment distance. The bow shock also displays an apparent thickening at lower supersonic Mach numbers. Shock waves are extremely thin; around the scale of the mean free path of air molecules (Ref. 5) and it is not possible for the flow upstream of a shock in supersonic flow to “know” about its existence. Therefore, this thickening is suspected to be a result of much lower grid resolution in the area of the bow shock and not as a representation of reality. At higher Mach numbers the shock is both stronger and closer to the cylinder surface where the grid has finer resolution.
Figure 20: Supersonic flowfields for the circular cylinder.
Plotting the shock detachment distance measured from contour plots with experimental data from Alperin, Figure 21, shows a Mach number dependent trend that is a close resemblance to experimental trends. The coarse mesh appears to display the best agreement to experiment, however, all three are high at low Mach numbers and too low at higher Mach numbers.

The distances given are in terms of the cylinder diameter. Detachment distance was measured directly from a contour plot of density in the flowfield visually from the forward surface of the cylinder to a location in the approximate center of the shock gradient. Combining numerical results along with experiment shows that the detachment distance reaches infinity as the freestream Mach number approaches one. Figure 20a shows no visible shock for this reason. The grid used is limited in its extent upstream of the cylinder and therefore bow shocks will not be visible with this particular grid size when the freestream Mach number is less than approximately 1.1, (Ref. 9).

One final notable observation in the contours in Figure 20 are strange patterns in the flowfields. These patterns were noticed for all supersonic solutions and all grids. The symmetrical grid was extended further out in terms of the cylinder diameter to find out if this was something real. Altering the grid size showed that the diamond-shaped “reflections” in the contours did not appear at the same relative locations from the cylinder body. Instead they were in the same relative location in the domain and appear to be an effect of the domain boundaries (Figure 22). Note, the bottom portions of the flowfield are omitted. This is an example of another phenomena of the solver/domain that the CFD user must recognize as not apart of a real flowfield. The effect of the
pattern is unknown but given the shallow gradients involved it is expected to be very minor. The interaction of the pattern with the farfield boundary as a “reflection” implies that this pattern will be found further downstream in solutions computed using even larger grid domains.

![Mach 1.3 density contours. Note relative size of the cylinder to that of the domain.](image)

Figure 22: Mach 1.3 density contours. Note relative size of the cylinder to that of the domain.

### 3.1.3 Normal Shock Relations

The leading edge of the bow shock is perpendicular to the flow providing an additional way for the CFD user to validate their results. Compressible flow theory gives a theoretical foundation for the flow conditions immediately downstream of a normal shock based on the freestream conditions. It can be shown that these thermodynamic relations can be found simply from the Mach number. It is left for the reader to consult the literature for the derivation of the shock relations. For three of the four relations, pressure, density, and temperature, the formulae are given as a ratio of the downstream to that of the upstream value. Mach number is the exception. The four normal shock relations are given by:
\[
\frac{P_2}{P_1} = 1 + \frac{2\gamma}{\gamma + 1} \left(\frac{M_i^2}{1}\right) - 1 \\
\frac{\rho_2}{\rho_1} = \frac{M_i^2}{1 + \frac{\gamma - 1}{\gamma + 1} \left(\frac{M_i^2}{1}\right) - 1} \\
\frac{T_2}{T_1} = \left[1 + \frac{2\gamma}{\gamma + 1} \left(\frac{M_i^2}{1}\right) - 1\right] \frac{2 + (\gamma - 1)M_i^2}{(\gamma + 1)\left(\frac{M_i^2}{1}\right) - 1} \\
M_2^2 = \frac{1 + \frac{\gamma - 1}{\gamma} \left(\frac{M_i^2}{1}\right)}{\gamma - \left(\frac{\gamma - 1}{\gamma}\right)}
\]

Computational values for the normal shock relations were obtained from contour plots of each of the respective parameters. As in the distance of the bow shock detachment, there was difficulty in locating a position that could be considered immediately downstream of the shock. In real flows, the shock is extremely thin with changes occurring almost instantaneously. Because of the finite cell size in the CFD domain, the shocks appear to have an effect on the upstream flow. This gives them a thickness that makes locating the shock difficult. Figure 23 is a plot of all four theoretical shock relations with measured computational values of density, Mach number, and temperature for the three supersonic cases examined.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{shock_relations.png}
\caption{Comparison of computed normal shock relations to theory: Density (○), Mach No. (□), Temperature (△).}
\end{figure}

The difficulty in measuring the relative values of pressure led to significant uncertainties. The scale of this uncertainty is such that the data obtained provides no useful information regarding computational accuracy. Only three computed relations,
density, Mach number, and temperature, are included here. It is clear that had the pressure values been more easily obtainable, their agreement with theory is expected to be similar to the other three parameters. Density ratio measurements are slightly lower than theory with divergence at the upper limits of the solution Mach number. The spread out shocks, as explained above, made locating the shock difficult at best, however all measurements were taken in the same location for each Mach number. In the end, the density discrepancy could not be fully explained.

3.2 NACA 0012

3.2.1 Flow Coefficients

The major flow coefficients, lift, drag, and moment, are important for airfoil analysis as applied to aircraft design. Here only the lift and drag are examined. For a symmetrical airfoil, the moment coefficient will be essentially zero, (Ref. 1). What is of concern here is how those coefficients change as a result of compressibility. Table 4 shows the lift coefficients for all four subsonic Mach numbers examined for each of the grids used.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.226</td>
<td>0.227</td>
<td>.229</td>
</tr>
<tr>
<td>0.5</td>
<td>0.249</td>
<td>0.251</td>
<td>.252</td>
</tr>
<tr>
<td>0.75</td>
<td>0.350</td>
<td>0.351</td>
<td>.358</td>
</tr>
<tr>
<td>0.9</td>
<td>0.149</td>
<td>0.191</td>
<td>.162</td>
</tr>
</tbody>
</table>

Table 4: Airfoil calculated lift coefficients.

There is very close agreement among all three grids showing little dependence except at $M = 0.9$ mirroring what was observed in the cylinder solutions. The reason for this is unclear; however, multiple shocks are present at subsonic Mach numbers this high. There is also a significant area of separated flow at this velocity just aft of the upper shock, which would be highly grid dependent due to resolution of the eddy scale, (Ref. 25). The variation of the lift coefficient follows that of experiment, (Ref. 1), with a drop in lift after $M = 0.75$. Unfortunately there was an insufficient variety of Mach numbers examined to find the exact location of the minimum and maximum and compare well to experiment.

The Reynolds number for $M = 0.3$ is approximately 9x10$^6$. Abbott and VonDenhoff provide data for this Reynolds number, which closely matches the computational results to within the readability of the plot on page 462, (Ref. 1). The cylinder results showed very measureable effects as a result of compressibility in the slightly high stagnation pressure coefficient. The lift coefficient for the NACA 0012 airfoil appears to show little, if any, compressibility effects.

Figure 24 shows the flowfields around the airfoil at all four Mach numbers. The grid for the figure is the fine grid. The lowest two Mach numbers show little highlights
or interesting flow characteristics. There is the leading edge stagnation point and an area
of flow disruption just past the half-chord point. The area appears to not be large enough
to exhibit flow separation that shouldn’t be expected at such a small angle of attack as
this.

Reaching $M = 0.75$ brings supercritical flow and the appearance of a
recompression shock on the upper surface. This shock is seen by the sharp gradient at
about $0.45c$. The shock also brings an increase in the area of disturbed flow just behind
the shock. The effect of the shock is discussed later. At $M = 0.9$ a recompression shock
on the lower surface is now present at the trailing edge. Significantly retarded flow as a
result of the strong upper surface shock means that the flow along the lower surface must
be slowed in order to produce a smaller velocity gradient at the trailing edge. Overall, the
flow over both surfaces of the airfoil is fairly smooth. The lack of large flow disruptions
is also a reflection of the increase in the lift coefficient seen in the literature and should
be present in the pressure distribution. These figures should be compared to the diagrams
shown in Figure 3.
Pressure coefficient plots are given in Figure 25 for each of the subsonic Mach numbers with experimental values plotted along with the CFD results for all except $M = 0.9$. The CFD results appear to not agree well with experiment, (Ref. 12), however, it should be noted that no effort was made by Ladson, et. al. to correct for experimental wall effects and the results are considered sufficient for tunnel comparison purposes. Additional experimental data set from McDevitt and Okuno, (Ref. 15), are included for $M = 0.75$. Here the authors consider the results at this particular Reynolds number to be sufficient for code validation. The excellent agreement the CFD solutions have with their experimental results reflect this assertion.

For $M = 0.3$ and 0.5, the pressure distribution is very smooth as would be expected for subcritical freestream velocities (Ref. 12). Reaching $M = 0.75$ brings the strong discontinuity caused by the presence of the shock on the upper surface. Mach 0.9 brings back the largely smooth pressure distribution mentioned above except for the upper and lower surface shocks placed well aft, near the trailing edge. The far aft shocks imply a lack of significant flow separation and possibly lower pressure drag. If this is the case, then the drag coefficient should reflect this. Drag coefficient values are shown in Table 5.
$M = 0.3$

$M = 0.5$
Figure 25: Subsonic airfoil pressure distributions with experiment. Note, no experimental data for $M = 0.9$. 

$M = 0.75$ 

$M = 0.9$
Table 5: Airfoil calculated drag coefficients.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.00881</td>
<td>0.00868</td>
<td>0.0083</td>
</tr>
<tr>
<td>0.5</td>
<td>0.00844</td>
<td>0.0083</td>
<td>0.00793</td>
</tr>
<tr>
<td>0.75</td>
<td>0.0175</td>
<td>0.0176</td>
<td>0.0169</td>
</tr>
<tr>
<td>0.9</td>
<td>0.103</td>
<td>0.123</td>
<td>0.116</td>
</tr>
</tbody>
</table>

The drag coefficient remains fairly flat until the flow is above the critical velocity. The drag doubles by $M = 0.75$ then increases even more dramatically by $M = 0.9$. However, this trend is not reflected in the experimental data, Ref. 1. Experiment shows that the drag increases exponentially towards sonic freestream velocities. A decrease in the pressure component of drag should be reflected by a lower drag value. The discrepancy with experiment is not explained at this time.

The lift for $M = 0.3$ show good agreement with experiment, (Ref. 1), however, there is significant error in the drag coefficient for this Reynolds number of $C_d = 0.006$. The literature gives compressibility as a possible explanation. It is stated by Abbott and VonDenhoff that compressibility will initially appear as an increase in drag. Experimental figures used for comparison here are given for incompressible flows and high Reynolds numbers. The conclusion that can be drawn from this is that compressibility effects are present even at Mach numbers as low as 0.3. Drag studies should not ignore compressibility, however, for lift it may be ignored until the freestream Mach number approaches critical.

3.2.2 Grid Dependence and Adaptation

The most significant grid dependence that is to be expected in this study should come in the higher transonic Mach numbers around areas of shock formation. This arises from how the solver is able to deal with the sudden changes in flow properties across the shocks and shock-boundary layer interaction. The retarded flow in the boundary layer as well as the effects outside of the boundary layer make this less of a trivial matter than the rest of the flowfield already is. The comparison with experimental data shown above in Figure 25 shows that a finite slope to the pressure gradient across the shock exists. This is as expected within the boundary layer where the shock becomes smeared in appearance.

Figure 26 shows the same experimental data along with pressure coefficient values from three CFD solutions (from each of the three different grids) plotted together for $M = 0.75$. While the overall shape of the two come remarkably close, the area immediately around the shock shows a difference that must be examined more closely. The plots in Figure 25 are from the finest grids, however, Figure 26 includes this data along with the pressure coefficient results for two coarser grids. The CFD results show little dependence on the grid for much of the span of the airfoil. As in the study by
Scalabrin and Azevedo, (Ref. 24), there was significant downward overshoot immediately aft of the shock that was remedied by increased grid resolution. The increase in grid resolution for this study also decreased the overshoot such that with the fine grid, there is almost no overshoot aft of the shock. What isn’t presented in Refs. 3 and 23 is the increase in the shock’s pressure slope which still deviates from experiment. A sharp slope implies a lack of shock-boundary layer interaction, which will broaden the pressure change across the shock (Ref. 23). An examination of a vector plot displays the degree to which the boundary layer is resolved (Figure 27). The resolution in the flow direction may be satisfactory however; the location of the first grid point might be at a distance that is too far from the surface. The magnitude of $y^+$ may be too great for to properly predict the boundary layer around the flow. Variation of the boundary layer resolution normal to the airfoil surface is left for further study.

![Figure 26: Grid dependence of pressure coefficient.](image-url)
Figure 27: Vector and contour plots showing the shock and boundary layer.

For all the cases shown in Figure 26, the resolution of the grid over the entire computational domain was varied. For the study by Scalabrin and Azevedo, the grid was
refined in areas with larger density variations using a density “sensor” which set a threshold for refinement based on density variation. FLUENT allows the user to operate in a similar manner using the results of a previously computed solution in the same manner as Scalabrin and Azevedo. Using density contours from a solution of 1600 iterations as the source of the gradient information, the medium grid was refined based on pressure coefficient contour gradients. The contours allow refinement in the area of the shock without excess refinement in places like the boundary layer and the wake downstream of the airfoil. A CFD student should note that proper refinement should still include the entire boundary layer and wake regions excluded to ensure more accurate force values and discrepancies found at lower Mach numbers.

Figures 26 and 27 show the results of the refinement to the medium resolution grid along with illustrations of the grid before and after refinement. The refinement clearly resulted in surface pressures in better agreement with the unrefined grids shown above.

![Figure 28: Comparison of refined grid to experiment.](image)

Figure 29: Grid changes as a results of refinement.
With the gradient refinement the changes are localized to where the largest flowfield changes occur. In the grid illustrations above, the shock location is visible by the strong cell concentrations. Refinement was made in three steps to properly resolve the shock gradient and other important features. An example of the advantage of gradient refinement is in the cell counts. For the original medium grid, the total cell count is 40,488 with the refined grid having 54,945 cells. Compare this to the more than 120,000 cells in the fine grid that displays no significant increase in accuracy over the refined grid. It can even be argued that the refined grid shows slightly better agreement with experiment in the vicinity of the normal shock. This along with the significant reduction in computational resources makes localized grid refinement a more tempting choice over a completely new grid with a very high cell count. Further refinement at the shock and in the boundary layer should bring future computational results to a virtual match of experimental results.
Chapter 4 Conclusion

Transonic and supersonic flow problems can be an excellent tool for new users of CFD to experience many aspects of flow simulation. The ability to represent shocks and associated flow changes also makes it a great visualization tool for students. The various effects aren’t going to all be seen in one simple case but by exploring both blunt and streamlined bodies, it was possible to work on many of them. While some of the results may leave the reader with questions regarding comparisons of results to experiment, it is only fair to make the comparison of CFD technique and strategy to similar CFD problems in the literature. Simulations by Squires, et. al. and Scalabrin and Azevedo show that the solution methods used here in the context of a student user in computation fluids were successful and valid. This does not mean that this is a tutorial in advanced CFD methods but as an introduction to simulation strategies.

Given that, the cylinder produced the most difficult flowfield to solve given its blunt geometry and large turbulent wake region. Convergence proved to be tricky in that techniques used to stabilize the flow slowed convergence. This in turn mimicked convergence when monitoring force values. Observance of the bow shock migration in the supersonic flow solutions displays the necessity to monitor multiple items when determining convergence. Even once the solution has reached convergence, the examination of surface pressure coefficients show that convergence and accuracy are indeed independent of each other. This should mean to the new CFD user that an accurate solution will not be found without proper convergence; however, a properly converged solution does not imply solution accuracy. Both the grid dependence and the inaccurate force coefficients found in the initial solutions of some Mach numbers disappeared when a second-order solution scheme was used. Second-order schemes in the solution controls brought separate first-order grid dependent solutions to much closer agreement. Dependence was still very apparent in regions where meshing just outside of the boundary layer was much different.

Simulation of the NACA 0012 airfoil as a classic example of a test case displayed very useful grid dependence around the area of the shock. This mirrors what was found in the literature in which shock-boundary layer interaction proved to be sensitive to grid resolution. The effect found both in the literature and in this study include inaccurate pressure across the shock and overshoot of the drop immediately downstream. This case was true grid dependence, unlike that mentioned above for the cylinder, since all airfoil flow simulations were completed using under the same solution control settings with the addition of monitoring lift coefficient. Transonic flow simulations should be regarded as both useful to the CFD student in both the practical sense and in its relevance to current design efforts around the world. From the author’s experience in solving the problems presented here and the observation in the solutions, teaching material was created (Appendix B and C). This material is appropriate for an introductory class in CFD and associated techniques. It is will also give students a chance to explore transonic and supersonic flowfields under different conditions.
REFERENCES


Ref. 18 Muse, T.C., Some Effects of Reynolds and Mach Numbers on the Lift of an NACA 0012 Rectangular Wing in the NACA 19-Foot Pressure Tunnel, Bulletin 3E29, NACA, 1943.


### APPENDIX A  Airfoil Coordinates

<table>
<thead>
<tr>
<th></th>
<th>Upper Surface</th>
<th></th>
<th>Lower Surface</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.0003</td>
<td>0.003</td>
<td>0.0003</td>
<td>0.003</td>
</tr>
<tr>
<td>0.0011</td>
<td>0.0059</td>
<td>0.011</td>
<td>0.0011</td>
<td>0.0059</td>
</tr>
<tr>
<td>0.0025</td>
<td>0.0088</td>
<td>0.0025</td>
<td>0.0025</td>
<td>0.0088</td>
</tr>
<tr>
<td>0.0045</td>
<td>0.0116</td>
<td>0.0045</td>
<td>0.0116</td>
<td>0.0045</td>
</tr>
<tr>
<td>0.0071</td>
<td>0.0144</td>
<td>0.0071</td>
<td>0.0144</td>
<td>0.0071</td>
</tr>
<tr>
<td>0.0102</td>
<td>0.0172</td>
<td>0.0102</td>
<td>0.0172</td>
<td>0.0172</td>
</tr>
<tr>
<td>0.0138</td>
<td>0.0199</td>
<td>0.0138</td>
<td>0.0199</td>
<td>0.0199</td>
</tr>
<tr>
<td>0.0181</td>
<td>0.0225</td>
<td>0.0181</td>
<td>0.0225</td>
<td>0.0225</td>
</tr>
<tr>
<td>0.0229</td>
<td>0.0251</td>
<td>0.0229</td>
<td>0.0251</td>
<td>0.0251</td>
</tr>
<tr>
<td>0.0282</td>
<td>0.0276</td>
<td>0.0282</td>
<td>0.0276</td>
<td>0.0276</td>
</tr>
<tr>
<td>0.0341</td>
<td>0.0301</td>
<td>0.0341</td>
<td>0.0301</td>
<td>0.0301</td>
</tr>
<tr>
<td>0.0405</td>
<td>0.0325</td>
<td>0.0405</td>
<td>0.0325</td>
<td>0.0325</td>
</tr>
<tr>
<td>0.0475</td>
<td>0.0348</td>
<td>0.0475</td>
<td>0.0348</td>
<td>0.0348</td>
</tr>
<tr>
<td>0.055</td>
<td>0.037</td>
<td>0.055</td>
<td>0.037</td>
<td>0.037</td>
</tr>
<tr>
<td>0.0631</td>
<td>0.0392</td>
<td>0.0631</td>
<td>0.0392</td>
<td>0.0392</td>
</tr>
<tr>
<td>0.0716</td>
<td>0.0412</td>
<td>0.0716</td>
<td>0.0412</td>
<td>0.0412</td>
</tr>
<tr>
<td>0.0807</td>
<td>0.0432</td>
<td>0.0807</td>
<td>0.0432</td>
<td>0.0432</td>
</tr>
<tr>
<td>0.0904</td>
<td>0.0451</td>
<td>0.0904</td>
<td>0.0451</td>
<td>0.0451</td>
</tr>
<tr>
<td>0.1005</td>
<td>0.0469</td>
<td>0.1005</td>
<td>0.0469</td>
<td>0.0469</td>
</tr>
<tr>
<td>0.1112</td>
<td>0.0486</td>
<td>0.1112</td>
<td>0.0486</td>
<td>0.0486</td>
</tr>
<tr>
<td>0.1223</td>
<td>0.0502</td>
<td>0.1223</td>
<td>0.0502</td>
<td>0.0502</td>
</tr>
<tr>
<td>0.134</td>
<td>0.0517</td>
<td>0.134</td>
<td>0.0517</td>
<td>0.0517</td>
</tr>
<tr>
<td>0.1461</td>
<td>0.053</td>
<td>0.1461</td>
<td>0.053</td>
<td>0.053</td>
</tr>
<tr>
<td>0.1587</td>
<td>0.0543</td>
<td>0.1587</td>
<td>0.0543</td>
<td>0.0543</td>
</tr>
<tr>
<td>0.1719</td>
<td>0.0554</td>
<td>0.1719</td>
<td>0.0554</td>
<td>0.0554</td>
</tr>
<tr>
<td>0.1854</td>
<td>0.0565</td>
<td>0.1854</td>
<td>0.0565</td>
<td>0.0565</td>
</tr>
<tr>
<td>0.1995</td>
<td>0.0573</td>
<td>0.1995</td>
<td>0.0573</td>
<td>0.0573</td>
</tr>
<tr>
<td>0.2139</td>
<td>0.0581</td>
<td>0.2139</td>
<td>0.0581</td>
<td>0.0581</td>
</tr>
<tr>
<td>0.2289</td>
<td>0.0587</td>
<td>0.2289</td>
<td>0.0587</td>
<td>0.0587</td>
</tr>
<tr>
<td>0.2443</td>
<td>0.0593</td>
<td>0.2443</td>
<td>0.0593</td>
<td>0.0593</td>
</tr>
<tr>
<td>0.26</td>
<td>0.0596</td>
<td>0.26</td>
<td>0.0596</td>
<td>0.0596</td>
</tr>
<tr>
<td>0.2763</td>
<td>0.0599</td>
<td>0.2763</td>
<td>0.0599</td>
<td>0.0599</td>
</tr>
<tr>
<td>0.2929</td>
<td>0.06</td>
<td>0.2929</td>
<td>0.06</td>
<td>0.06</td>
</tr>
<tr>
<td>0.3099</td>
<td>0.06</td>
<td>0.3099</td>
<td>0.06</td>
<td>0.06</td>
</tr>
<tr>
<td>0.3273</td>
<td>0.0598</td>
<td>0.3273</td>
<td>0.0598</td>
<td>0.0598</td>
</tr>
<tr>
<td>0.3451</td>
<td>0.0596</td>
<td>0.3451</td>
<td>0.0596</td>
<td>0.0596</td>
</tr>
<tr>
<td>0.3633</td>
<td>0.0592</td>
<td>0.3633</td>
<td>0.0592</td>
<td>0.0592</td>
</tr>
<tr>
<td>0.3818</td>
<td>0.0586</td>
<td>0.3818</td>
<td>0.0586</td>
<td>0.0586</td>
</tr>
<tr>
<td>0.4007</td>
<td>0.058</td>
<td>0.4007</td>
<td>0.058</td>
<td>0.058</td>
</tr>
<tr>
<td>0.4199</td>
<td>0.0572</td>
<td>0.4199</td>
<td>0.0572</td>
<td>0.0572</td>
</tr>
<tr>
<td>0.4395</td>
<td>0.0563</td>
<td>0.4395</td>
<td>0.0563</td>
<td>0.0563</td>
</tr>
<tr>
<td>0.4594</td>
<td>0.0553</td>
<td>0.4594</td>
<td>0.0553</td>
<td>0.0553</td>
</tr>
<tr>
<td>0.4795</td>
<td>0.0541</td>
<td>0.4795</td>
<td>0.0541</td>
<td>0.0541</td>
</tr>
<tr>
<td>0.5</td>
<td>0.0529</td>
<td>0.5</td>
<td>0.0529</td>
<td>0.0529</td>
</tr>
</tbody>
</table>
MATLAB Function

```matlab
function naca(des, num, open)

a0= 0.2969;            %Declare coefficients
a1= -0.1260;
a2= -0.3516;
a3= 0.2843;

if open ==1
    a4= -0.1015;  % Open TE
else
    a4= -0.1036;  % Closed TE
end

deg = linspace(0,pi/2,num);

x = -(cos(deg)-1);         %Create x-coordinates

camb=floor(des/1000);

perc = floor((des-camb*1000)/100);

t = floor(((des-camb*1000)-perc*100))/100;

m = camb/100;           %Maximum camber
p = perc/10;            %Max camber location

y1 = t./2*(a0*x.^(1/2)+a1*x+a2*x.^2+a3*x.^3+a4*x.^4); %Calculate surface
```

<table>
<thead>
<tr>
<th>x</th>
<th>y1</th>
<th>x</th>
<th>y1</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5208</td>
<td>0.0515</td>
<td>0.5208</td>
<td>0.0515</td>
</tr>
<tr>
<td>0.5418</td>
<td>0.05</td>
<td>0.5418</td>
<td>0.05</td>
</tr>
<tr>
<td>0.5631</td>
<td>0.0484</td>
<td>0.5631</td>
<td>0.0484</td>
</tr>
<tr>
<td>0.5846</td>
<td>0.0467</td>
<td>0.5846</td>
<td>0.0467</td>
</tr>
<tr>
<td>0.6063</td>
<td>0.0449</td>
<td>0.6063</td>
<td>0.0449</td>
</tr>
<tr>
<td>0.6283</td>
<td>0.043</td>
<td>0.6283</td>
<td>0.043</td>
</tr>
<tr>
<td>0.6505</td>
<td>0.0411</td>
<td>0.6505</td>
<td>0.0411</td>
</tr>
<tr>
<td>0.6729</td>
<td>0.039</td>
<td>0.6729</td>
<td>0.039</td>
</tr>
<tr>
<td>0.6955</td>
<td>0.0368</td>
<td>0.6955</td>
<td>0.0368</td>
</tr>
<tr>
<td>0.7183</td>
<td>0.0345</td>
<td>0.7183</td>
<td>0.0345</td>
</tr>
<tr>
<td>0.7412</td>
<td>0.0321</td>
<td>0.7412</td>
<td>0.0321</td>
</tr>
<tr>
<td>0.7642</td>
<td>0.0297</td>
<td>0.7642</td>
<td>0.0297</td>
</tr>
<tr>
<td>0.7874</td>
<td>0.0271</td>
<td>0.7874</td>
<td>0.0271</td>
</tr>
<tr>
<td>0.8107</td>
<td>0.0245</td>
<td>0.8107</td>
<td>0.0245</td>
</tr>
<tr>
<td>0.8342</td>
<td>0.0218</td>
<td>0.8342</td>
<td>0.0218</td>
</tr>
<tr>
<td>0.8577</td>
<td>0.0189</td>
<td>0.8577</td>
<td>0.0189</td>
</tr>
<tr>
<td>0.8813</td>
<td>0.016</td>
<td>0.8813</td>
<td>0.016</td>
</tr>
<tr>
<td>0.9049</td>
<td>0.013</td>
<td>0.9049</td>
<td>0.013</td>
</tr>
<tr>
<td>0.9287</td>
<td>0.0099</td>
<td>0.9287</td>
<td>0.0099</td>
</tr>
<tr>
<td>0.9524</td>
<td>0.0067</td>
<td>0.9524</td>
<td>0.0067</td>
</tr>
<tr>
<td>0.9762</td>
<td>0.0034</td>
<td>0.9762</td>
<td>0.0034</td>
</tr>
</tbody>
</table>

1 0 1 0
cline = zeros(1,num);
coor=zeros(num,2);

for n = 2:num
    if x(n)<=p
        cline(n)= m*x(n)/p^2*(2*p-x(n));           %Create camber line
    else
        cline(n)=m*(1-x(n))/(1-p)^2*(1+x(n)-2*p);
    end
end

y=y1+cline;                %Add camber to upper surface
ybot=cline-y1;              %Build lower surface

plot(x,y);
hold on
plot(x,cline,'r');         
plot(x,ybot);

coor(:,1)=x';
coor(:,2)=y';
coor
APPENDIX B  Fluent Transonic Airfoil Tutorial

INTRODUCTION

This tutorial is meant as a step-by-step explanation of the process required to model the transonic flow around an airfoil shape. This complex problem presents a number of difficult factors to simulate. These include compressible flow, turbulent boundary layers, flow separation, and shocks. The solution process is laid out along with explanation. The first example uses a simple, coarse airfoil grid to speed up the solution process and give the reader an introduction to simulation and post processing. Additional grids are included with finer grid resolutions for the student to explore additional topics in CFD.

PROBLEM SETUP

Start Fluent. A small window opens to select the dimensions and precision needed. This problem is two-dimensional and double precision is desired. Select 2ddp and continue to the home screen by clicking Run.

Once Fluent is executed, the window should appear similar to one below. While the interface for Fluent is graphical in nature, output is in a text format. In this window, results and data produced during execution will appear.
The first step to simulation of transonic flow is to open the grid in question. Files with the extension .MSH are Fluent-specific grid files. Another file type discussed later has the extension .CAS. This is a data file for a specific case that was previously run and saved. Case files give the user the ability to come back later and examine results, correct for mistakes or crashes, or to continue iterating if necessary.

- **Click on File, Read, Case.** From here find and click on the file titled NACA0012.msh.

Once opened Fluent will do a series of checks and provide the user with statistics concerning the grid. This includes the numbers of cells, nodes, and minimum and maximum cell sizes. Upon opening the NACA0012.msh file, the following screen should be visible.

Grid quality is fundamental to an accurate solution. While Fluent cannot determine if the grid is optimal, it does include a function to examine the grid for consistency and obvious problems that will hamper efforts. The check displays the dimensions in the selected units, the cell volumes, and other statistics. It is important to look over the minimum and maximum volumes. Volumes should be reasonable based on the size of the grid and, most importantly, the minimum volume should be greater than zero; no negative volumes!

- **To check the grid if Fluent, click Grid, Check.** The following screen should be visible.
One can view the actual grid and explore it in Fluent.

- **The grid can be viewed by clicking on **Display, Grid. This opens a dialog window showing options for viewing the grid.**
- **Click Display.**

From here the user can zoom in and pan around to see how the grid is constructed.

- To zoom in, use the button in your mouse’s scrolling wheel to draw a box around the area of interest starting from the upper left and moving down towards the lower right. To zoom out, draw the box from the lower right to the upper left. Use the left mouse button to click and drag the image round.
There are a few features that should be noted in the grid. These are important to accurate modeling of the physics involved. Zooming in close to the edges of the airfoil shows the concentrated grid cells. These are necessary to resolve the flow near the wall. Since this is a viscous case with turbulence, there will be significant flow changes near the walls.

For viscous flows, it is very important to keep in mind the presence of the boundary layer. The grid must be able to resolve both the large-scale and small-scale flow characteristics if one is to expect accurate results. It is common practice to include 6-8 node points within the boundary layer.

Zooming in to the area aft of the trailing edge shows finer grid cells well beyond the location of the boundary layer. With the turbulent models used and the presence of the shock, separated flow is expected with a large wake downstream. The grid has been made finer along the chord axis downstream of the trailing edge to resolve the wake properly.
GRID SCALING

The NACA 0012 grid was created with a chord length of one meter. For this problem the Reynolds number is desired to be $10^7$; however, it is also desired that the Mach number be set to a known value of 0.75. Given the formula

$$\text{Re} = \frac{\rho v L}{\mu}$$

and the fixed fluid properties, the only way to set the Reynolds number to that desired for our problem is to rescale the grid such that the chordlength is the correct value. Solving for $L$ gives a chordlength of 56.2 centimeters.

- To rescale the grid click Grid, and Scale.
- In the X and Y fields enter 0.562.
- Click Scale. The parameters showing the extent of the domain should now change to 56.2% of their initial values. Note, the figure below shows the dialog screen before clicking Scale.
- Click Close.

MODELS

An airfoil in transonic flow operates at high Reynold’s Number so turbulent flow is expected for all cases. For higher subsonic speeds, normal shocks can be expected along with flow separation and possible recirculation. This means that turbulence should be turned on if one expects decent quantitative results.

- Clicking on Define, Models, Viscous brings up the flowing window.
For this case, the Spalart-Allmaras model will be used. From the Fluent documentation:

The Spalart-Allmaras model is a simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed for aerospace applications involving wall-bounded flows and has shown to give good results for boundary layers subjected to adverse pressure gradients.

One advantage in using the Spallart-Allmaras model is in computation time. With only one equation to work with, compared to two or five seen in others, computation time is reduced dramatically.

• Enable the model by clicking on the radio button next to “Spallart-Allmaras.”

The window changes, showing the model options.

• Leave all defaults as is and click OK.

MATERIALS

The working fluid for this problem will be air. However, since the conditions are known to be transonic in nature, compressibility must be taken into account, as well as the effects of temperature changes the fluid will be experiencing.
• Click **Define, Materials** to open the Material option window.
• In the Density drop down list, select “Ideal-gas.”
• In the Viscosity drop-down list select Sutherland. Another window will open up with Sutherland Law options.
• Click **OK** to accept the default options.
• Click **Change/Create**, then **Close**.

**BOUNDARY CONDITIONS**

For this problem there are only two boundaries that are of interest. The first is the surface of the airfoil. The second is the outer boundaries of the grid. It is important to begin the solution with the flow set as close to the actual conditions as possible by setting proper velocity, pressure, and turbulence quantities to the flowfield. The closer one starts to the actual flow, the quicker the solution will converge. This will also ease convergence in situations where quickly changing flow properties lead to divergence.

• **Open the boundary conditions window by clicking Define, Boundary Conditions.**
On the left will be a list of four zones. These are airfoil, farfield, default-interior, and fluid. It is only known what the freestream conditions are.

- **Click and highlight farfield.** In the right pane of the box shows a list of boundary types. The farfield boundary should be listed as “pressure-far-field.”
- **Click Set.**

![Pressure Far-Field](image)

As mentioned in the grid scaling procedure, the desired flow conditions are sea level standard, Mach 0.75, angle of attack of two degrees, and freestream Reynolds number (chord based) of $10^7$. Much study has been done in this area and a large amount of experimental data exists with which to confirm results.

- **Next to Mach Number, enter 0.75.**

Unfortunately, Fluent doesn’t allow one to simply enter the angle of attack in degrees. Instead, the user enters the x- and y-components of the flow. For the x-direction, the flow component is $0.999391$ ($x = \sin(2)$). The y-component is $0.034899$ ($y = \cos(2)$).

- **Enter this next to X-Component of Flow Direction. Enter the Y-component in the next line down.**

It is possible to take angle of attack into account when the grid is generated by tilting the airfoil geometry. However, this grid is intended for use in multiple cases in which the flow may be at different incidence angles.

- **At the bottom select Turbulent Viscosity Ratio next to Specification Method and leave the default ratio at 10.**

If the turbulence is known to be of a different value, for example the entrance to a pipe, one can choose a different default ratio to speed up convergence.
• *Click OK and close the Boundary Conditions window.*

**SOLUTION CONTROLS**

• *Click Solve, Controls to open the Solution Controls window.*

The Solution Controls window allows the user great control over the iteration process and convergence speed. For many cases, the default values are excellent and provide quick convergence and stable solutions. However, for many types of flows, instability will be present. These instabilities can result from a number of sources. One common cause is a rapidly changing flowfield from unsteady flow and/or boundary conditions that are less than optimal. The use of these parameters will be discussed in a later tutorial in which a case is examined where convergence is a problem. For this case, the default values should be satisfactory and allows for a quick, stable solution.

• *Click OK to exit.*

**INITIALIZATION**
• **Initialize the flow clicking **Solve, Initialize.**

• **Select farfield in the Compute From drop-down list. The window should appear like the one below. If it doesn’t, close the window and go through the previous steps to find the problem.**

• **Click Close.**

**MONITORS**

During execution, convergence must be monitored and checked for problems that may lead to divergence or inaccurate results. Many problems can be caught here and the run halted while the problem is fixed. The two types of monitors that are of interest for this problem are Residuals and Forces.

![Residual Monitors Window](image)

- **Open the Residual Monitors window by clicking Solve, Monitors, Residual.**
- **Enable plot at the upper left to create a graph of the residuals during execution.**

Near the bottom is a list of each of the residuals calculated by Fluent for the given problem setup. We don’t want to judge convergence using the Residuals. The reason why is explained below.

- **For now, uncheck all the boxes under Check Convergence. You may need to scroll down to see all residuals depending on the turbulence model selected.**

If you notice during execution that the solution has converged, one of the checkboxes may have been missed. The window should look like the image below.

- **Click OK.**

Since the entire goal of modeling an airfoil is to calculate its lift, drag, and moment coefficients, monitoring residuals is often a less-than-trivial method of judging convergence. Convergence can often be determined when all residuals drop below $10^{-3}$. 

64
Some problems, however, won’t be properly converged at this point. Others may never reach $10^{-3}$, even after many thousands of iterations. What is convergence? Simply put, it is when a solution variable meets or closes in on one specific value. For the airfoil, we want lift and drag. Once plots of lift and drag have leveled off at a single value, it can be assumed with confidence that the solution has properly converged.

First step in enabling force monitoring is to tell Fluent what to calculate the forces from. Reference values provide Fluent the proper fluid characteristic and geometric sizing of the airfoil. From the formula for lift, $L = \frac{1}{2} \rho v^2 C_l S$, knowledge of the size of the airfoil is required for proper non-dimensionalization.

- **Clicking Report, Reference Values** brings up the reference values window.
- **In the Compute From drop down list, select far-field.**

The proper values for the fluid are automatically placed in the appropriate spaces. Earlier, the airfoil size was scaled in order to change the Reynold’s Number to the value desired for this problem. The original unscaled grid has a unit reference length and would require no change. The grid was scaled earlier to a shorter chordlength so now the reference values window may not have them correct to calculate the proper forces.

- **Change the values for Area (m$^2$) and Length (m) match those shown below.**

![Reference Values Window]

- Click **OK**.

Next, output of the force values must be enabled.

- **Click Solve, Monitors, Force** to open the Force Monitors window.
- **Enable Print and Plot.**
• Click and highlight the airfoil wall zone, set Plot Window to 1, and select Drag from the Coefficient drop-down list.
• Enter the force vector values $x = 0.999391$ and $y = 0.034899$. The window should look like the following.
• Click Apply.

![Force Monitor Window](image)

• For the lift coefficient, change the options to match those shown below.

![Force Monitor Window](image)

• Click Apply, then Close.

EXECUTION

The airfoil case has now been set up and is ready to be run.

• Click Solve, Iterate to bring up the Iterate window.
• Enter 100 for the Number of Iterations.
• Click Iterate to start the solution.

Immediately three windows should open up, Residuals, Drag Coefficient, and Lift Coefficient. In the main Fluent window, all plotted parameters are also output in raw form. Use the printout and plots to monitor convergence and stability of the solution. After 100 iterations have been completed, the residual and force plots should look like those shown in the figures below.
The fact that a solution has reached convergence is not an indication of solution accuracy. Examination of the results against experimentally obtained data is an indication of how well the solution agrees with a body in a real flowfield. Different methods exist for increasing solutions accuracy. They include modifying the grid to increase total resolution, dynamic grid adaptation to increase resolution around specific areas, second order discretization, and more. Since a converged solution has already been obtained for the NACA 0012 airfoil there are results to compare before and after.

The simplest method of increasing accuracy of a converged solution is to use Second-order Discretization. In the Solution Controls menu change Pressure to “Second Order” and Density, Momentum, Modified Turbulent Viscosity, and Energy to “Second-Order Upwind.” Continue iterating until convergence has been reached once more. Observe how changing to Second Order has affected the residual and force plots. How many iterations does it take to reach convergence? The next figure shows the Lift and Drag force plots after changing to Second order.
The original solution at 700 iterations showed lift and drag coefficients of 0.31 and 0.039, respectively. Six hundred additional iterations using Second-order discretization give 0.36 and 0.019. This shows that an accurate solution cannot be found without convergence, but at the same time, a converged solution does not imply an accurate one. Compare the results for lift and drag to experimental results of $C_l = 0.35$, and $C_d = 0.012$.

**POST PROCESSING**

Assuming that the solution has remained stable and good convergence has been reached, now the user can begin to analyze the results. There are a number of ways to examine a Fluent solution. The simplest is to output force values. These are coefficients of lift, drag, or moments that may be of interest to an engineer for either the design process or to verify the solution accuracy.

One of the most interesting and desirable is to create a contour drawing of the flow field. There are dozens of different contours to draw in Fluent. These can be pressure, density, turbulence, velocity, each with their own set of options. For example, pressure can be static, stagnation, or in coefficient form. Velocity can be absolute or by Mach number. Not only do contour plots provide great insight to the phenomena involved in the flow, they look pretty. The next figure shows the contour window in Fluent.

- **Click Display, Contours.**
• Change the options as seen in the figure to produce a contour of static pressure.

Below is what is shown when Display is clicked. At the right is a scale showing the full range of pressures in the flowfield. What is the maximum pressure? Where does it occur and how does this coincide with what is expected from theory?

Create contour plots of Mach number. What is the maximum Mach number and where does it occur? How does this agree with Bernoulli’s principles? Also, the asymmetry in the appearance of the Mach contours implies that the flow is accelerated along the upper surface compared to the lower surface. How would this appear if the angle of attack were zero? Hint: The NACA 0012 is a symmetrical airfoil. What would you expect the lift coefficient to be?

Fluent also has the ability to view the flowfield in the form of flow vectors. This function plots the flow as a set of vector arrows pointing in direction of the local flow and their lengths proportional to a desired parameter, for example, velocity. This is important in looking for proper boundary layer resolution and for areas of separated flow.

• Click Display, then Vectors to bring up the vector plot dialog.
• Select Vectors of Velocity, then Color by Velocity and Velocity Magnitude in the dropdown menus then click Display.
As in the contour plots, the entire grid will be shown in the window. Zoom in close to the airfoil to view the vector arrows. Zooming in close to the trailing edge (below) shows a region of highly retarded flow. Close to the wall the boundary layer should be visible. Does it appear that the boundary layer is fully or properly resolved? How does it change along the flow direction? Separated flow is visible as a reversal in the flow direction vectors (arrows pointing the wrong way). Are there any areas of flow separation? How would you expect this to change with increased angle of attack?

To find out exactly what is occurring around the airfoil, plots of surface pressure coefficient are very important to the design process. Fluent includes a function to produce and X-Y plot of this parameter. The dialog also includes a function output the data to a text file for importation into a spreadsheet program.
• **Click Plot**, and **XY Plot**.
• Select **Pressure and Pressure Coefficient** from the dropdown menus
• Select airfoil from the list of surfaces.
• **Click Display**.

Normally plots pf pressure coefficient appear reversed in the Y-direction. This places negative on the top and, for a cambered airfoil or one at a positive angle of attack, the uppermost data set corresponds to the upper surface. Since this is a transonic flow problem, how would a normal shock appear in a pressure coefficient plot? Are there any shocks present in this flowfield?

Now that there is knowledge of the solution process, go back and vary parameters like angle of attack and Mach number to see how the flowfield changes. Before doing so make sure to save your current solution case and data. With the new conditions, how does the iteration process change? Are there any conditions in which the solution
diverges? The next tutorial involving flow around a circular cylinder will explore methods of controlling the solution to avoid divergence and obtain solutions to more complex flows.

ADDITIONAL EXERCISES

- **Boundary Conditions:** Change the boundary conditions for the current problem setup. Changes can include changing the freestream Mach number and/or the angle of attack. What effects are seen? How far can they be pushed before convergence is not possible? What Mach number/angle of attack combinations produce critical flow?

- **Viscous Models:** Change viscous models to see what effects there are. Also try Laminar and Inviscid models. How does this affect the Lift and Drag results?

- **Grid Dependence:** Often the results and quality of a solution is dependent upon the grid itself. A very important test for a CFD simulation is to examine it for grid dependence. Other grids were included with this tutorial of differing resolutions. Solve for the original flowfield of M = 0.75 and angle of attack of two degrees. Is there any considerable difference that can be noted in lift and drag? How many iterations does it take to converge? Examine the boundary layer as well as plot flow vectors and pressure coefficient curves. Can a user conclude that the original case is grid dependent?

- **Critical Mach number:** Critical Mach number is defined as the freestream velocity at which the maximum surface speed of the fluid reaches Mach one. Run simulations at different Mach numbers and view the appropriate contour plot to see what the maximum Mach number is in the flowfield. Estimate the critical Mach number of the NACA 0012 airfoil at zero, two, and four degrees angle of attack.

- **Supercritical Airfoil:** Modern jet transports operate at freestream Mach numbers that would normally result in a classical airfoil operating well beyond its critical Mach number. Run a solution for the Supercritical mesh at multiple Mach numbers. Compare the flow characteristics and pressure coefficient to that of the NACA 0012 airfoil at the same Mach number and AOA. What characteristics are seen that would make supercritical airfoils more ideal for use at transonic freestream Mach numbers?
APPENDIX C  Fluent Transonic Cylinder Tutorial

INTRODUCTION

This tutorial is meant as a step-by-step explanation of the process required to model the supersonic flow around a circular cylinder using Ansys Fluent. This flowfield is difficult to simulate and often produces a divergent solution unless the user is knowledgeable about controlling progression of the solution. The problems in this tutorial include compressible flow, turbulent boundary layers, flow separation, shocks, and divergence. The solution process is laid out along with explanation. It is assumed that the reader has already gone through the NACA 0012 airfoil tutorial so detailed explanation of basic problem setup is skipped here.

PROBLEM SETUP

Problem setup is almost identical to that of the airfoil case. Start Fluent. Select 2ddp and continue to the home screen by clicking Run.

- Click on File, Read, Case. From here find and click on the file titled ______________.msh.
- Check the grid for consistency by clicking Grid, then Check.
- View the grid by clicking on Display, Grid. Leave everything at their default values and click Display. Examine the grid to see the different regions and cell distributions.
Be sure to zoom in close to the edges of the cylinder to see the concentrated grid cells. These are necessary to resolve the flow near the wall. Since this is a viscous case with turbulence, there will be significant flow changes near the walls. Viscosity will be turned on for this solution so a well-defined boundary layer is expected. Grid cell size is distributed such that cell size increases radially outward from the wall of the cylinder. Like that of the airfoil, the centerline portion of the downstream domain has increased resolution to account for the expected turbulent wake.

MODELS

Enable turbulent viscous flow by turning on the Spalart-Allmaras turbulence model.

- Click Define, Models, Viscous to bring up the flowing window.
  - Enable the model by clicking on the radio button next to “Spalart-Allmaras.”
  - Leave all defaults as is in the options menu and click OK.
MATERIALS

The working fluid for this problem will be air. Open the Materials dialog to enable compressible flow.

- Click Define, Materials to open the Material options window.
- In the Density drop down list, select “Ideal-gas.”
- In the Viscosity drop-down list select Sutherland. Another window will open up with Sutherland Law options.
- Click OK to accept the default options.
- Click Change/Create then Close.

BOUNDARY CONDITIONS

Set up the flowfield conditions for the domain through the Boundary Conditions menu.

- Click Define, Boundary Conditions.
The list of computational zones now include cylinder, default-interior, farfield, and fluid.

- Click and highlight farfield. In the right pane of the box shows a list of boundary types. The farfield boundary should be listed as “pressure-far-field.”
- Click Set.

As mentioned in the grid scaling procedure, the desired flow conditions are sea level standard and Mach 1.3. The cylinder is symmetrical in both x and y axes so the flow can have no angle of attack. Leave the X- and Y-Components at their default values.

- Next to Mach Number, enter 1.3.
- At the bottom select Turbulent Viscosity Ratio next to Turbulence Specification Method and leave the default ratio at 10.
- Click OK and close the Boundary Conditions window.

INITIALIZATIONS

- Initialize the flow clicking Solve, Initialize, Initialize.
- Select farfield in the Compute From drop-down list. The window should appear like the one below. If it doesn’t, close the window and go through the previous steps to find the problem.
• Click **Init**, then **Close**

**MONITORS**

As will be seen later, supersonic flow around a cylinder is very unstable and difficult to simulate. A bow shock will be present as well as flow separation leading into a large turbulent wake. Each of these point to possible difficulties for the solver. It is even more important to monitor the solution for stability to prevent divergence.

![Residual Monitors](Image)

- **Open the Residual Monitors window by clicking Solve, Monitors, Residual.**
- **Enable plot at the upper left to create a graph of the residuals during execution.**
- **Uncheck all the boxes under Check Convergence. You may need to scroll down to see all residuals for certain turbulence models.**
- **Click OK.**

As shown in the airfoil tutorial, convergence can often be determined when residuals drop below $10^{-3}$. Some problems, however, won’t be properly converged at this point. Others may never reach $10^{-3}$, even after many thousands of iterations. For the airfoil, lift and drag coefficients were monitored for convergence. A cylindrical geometry cannot produce a stable lift force without circulation. Drag is the dominant force and will be used to help determine convergence. The first step in enabling force monitoring is to tell Fluent what to calculate the forces from.

- **Clicking Report, Reference Values brings up the reference values window.**
- **In the Compute From drop down list, select far-field. The proper values for the fluid should be automatically placed in the appropriate spaces.**
- **The cylinder diameter in the grid is two meters. In the fields for Area and Length enter a value of 2.**
- **Click OK.**
Next, output of the force values must be enabled.

- **Click** Solve, **Monitors**, **Force** to open the Force Monitors window.
- **Enable** Print and Plot.
- **Click** and highlight the cylinder wall zone, set Plot Window to 1, and select Drag from the Coefficient drop-down list.
- **Leave** the force vector values at their defaults of 1 and 0. The window should look like the following.

```
• Click Apply, then Close.
```

**EXECUTION**

The cylinder case that has now been set up is ready to be run.

- **Click** Solve, **Iterate** to bring up the Iterate window.
- **Enter** 300 for the Number of Iterations.
• **Click Iterate to start.**

Immediately two windows should open up, Residuals and Drag Coefficient. In the main Fluent window, all plotted parameters are also output in raw form. Immediately once the solution begins to iterate, a major problem with the solution arises. The solution values for absolute pressure, temperature, and turbulent viscosity begin to go wild. In order to compensate, Fluent attempts to limit them in order to stabilize the solution. After five iterations, however, the solution diverges and Fluent halts the solution process. The home screen should look similar to the following.

![Home Screen Screenshot]

It is clear that steps have to be taken in order to control the solution more in order to achieve at least some results and to prevent divergence.
CONVERGENCE

For most relatively simple problems like the one examined in the airfoil tutorial, Fluent will converge with relative ease. These flowfields aren’t dominated by separation regions, bow shocks, or oscillating wakes. With the solution controls set to their default the solver only completed five or so iterations before temperature divergence was detected the solver. Much of the reason for the divergence is that flow development is too fast for the solver to handle and each iteration brings the solution further from the ideal result, also known as divergence. Three methods exist to remedy this. This first is to start the solution from a previously run result using more simple flow conditions. For example, running the solution to convergence with viscosity turned off, then enabling the appropriate viscous model and continuing to iterate. This can be successful, but experience has shown that it isn’t as trivial as one would expect. Often the inviscid solutions are just as unstable and will still require significant effort on the user’s part.

The second method is to turn off different equations. The Solution Controls menu contains the tools a user needs to change how the solver acts during iteration.

- **Click Solve, Controls, Solution Controls.** The window should look similar to the one in the figure below. Under Equations, there should be a list of two or more names highlighted. To turn off an equation, click on the name to deselect it. Do the opposite to turn that particular equation back on.

- For now turn off the Energy equations by clicking on it so that it is no longer highlighted. Reasons for turning off Energy will be explained later.
- Re-initialize the flow by clicking Solve, Initialize, Initialize.
- Select farfield from the dropdown list.
- Click Init. In the warning window click OK to discard the current values.
- Click OK.

Restart iterating. Immediately the same problem comes up with absolute pressure exceeding the solution limits. However, the number of cells that exceed the limit begins to decrease quickly. By nine iterations the limits are no longer exceeded and the solution
begins to stabilize. Allow the solver to continue until 100 iterations have been completed. View a contour plot of Velocity Magnitude. Do so every 100 iterations in order to get an idea of how the flowfield is developing. Below are four plots at 1, 25, 50, and 100 iterations. Notice that even in a time-independent solution, changes are present as if the flow was suddenly switched on. What features do you see appearing in the flow?

• Open the Solution Controls menu.
• Enable the Energy equation and click OK.
• Click Solution, Monitors, and Residual to open the solution monitors menu. Under Check Convergence, uncheck the box next to Energy.
• Continue the solution for another 200 iterations. Note: Do not initialize the flow again. Doing so will bring the solver back to iteration zero.

NOTE: Problems with different flow values can be dealt with by turning off the equations associated with them. For example, if it is found that the temperature is diverging at the beginning of the run, turn off the energy equation, reinitialize, and then restart the solution for 100-200 iterations. If pressure is exceeding the limits, turn off the Flow equation, etc.

After a short time the turbulent viscosity begins to exceed the limits in an ever-increasing number of cells. The appearance of the residual and drag plots should remain smooth and show no indication of divergence. These two coincidences can often be an indication that
the effect is a real one and not a result of an unstable solution. In other words, the Solution Limits are unnecessarily low.

- Click **Solve, Controls, Limits**.
- In the Solution Limits dialog, increase the Maximum Turbulent Viscosity Ratio to 1,000,000.
- Click **OK**.
- Continue iterating.

The limit problem should no longer be visible. Allow the solver to continue until the drag coefficient value printed out in the home screen stabilizes at or around a single value to three or four significant figures. For this simulation, the drag coefficient converged upon approximately $C_d = 1.549$. The maximum residual value is about $5 \times 10^{-4}$. Your result may be slightly different depending on exactly how many iterations the solution was allowed to run.

Below is a contour plot of velocity magnitude for the converged solution. The green hyperbolic shape is the result of a classic bow shock seen in front of all bodies moving at supersonic velocities. The shock itself is the leading edge of the green region. Also notice that there is no velocity gradient upstream of the bow shock, however, the velocity makes a sudden and immediate decrease across the shock as expected from normal shock theory. As a result of the supersonic flow velocity, the fluid upstream of the cylinder is “unaware” of its presence. The location of the shock, or detachment distance, is a function of the Mach number. Higher Mach numbers result in decreased detachment distance. This is left for the student to explore in a later section.

Another large feature visible below is the diamond-shaped pattern downstream of the wake. The upstream components are explored below. Further study using large domain regions shows the aft pattern to be a computational anomaly. The relative size of the diamond shape does not appear to change relative to the size of the whole grid. This implies that it may not be a component of a real flowfield and should, for the most part, ignored.
Before going on with additional solution strategies, zoom into the region immediately behind the cylinder. This is the most active region in the entire flowfield dominated by the large separated flow and turbulent wake. Here, specific compressible phenomena are clearly visible. The first are two diverging features emanating away from the trailing edge. These features, first mentioned above, are recompression shocks caused by the high velocity surface flow slowing down as it crests the upper and lower surfaces. Notice the higher flow velocity upstream of the shocks followed by an almost instantaneous decrease across them. What is the maximum flow velocity? How does this compare to that of the farfield boundary conditions?

The second notable feature is the symmetrical blue pattern immediately downstream of the wake is important. The colors indicate low flow velocities and highly retarded flow. To see what is really going on, create a vector plot of that region:

- Click **Display, Vectors**
- Select **Vectors Of Velocity, and Velocity Magnitude. Leave everything at their default values.**
- Click **OK.**
Zoom in and explore the area where the blue-colored meets the upper surface of the cylinder. The boundary layer shows textbook pinching and reversal an indication of flow separation. The lower portion of the boundary layer is now flowing upstream. Move the view over and down to see the centerline of the wake immediately behind the cylinder. Here, large symmetrical areas or circulation exist on the downstream side of the cylinder reversed flow along the centerline of the wake. The blue regions are the result of this circulation and a cause of a significant amount of pressure drag on the cylinder. Increasing the Vector Scale to 10 and Skip to 5 may help aid in visualizing what is going on.

UNDER-RELAXATION FACTORS

Not all cases can be successfully stabilized by turning equations off or increasing the solution limits. As seen above, some cases may be fixed by turning equations on and off. If doing this doesn’t quickly lead to a stable solution another effective method can achieve the same results. Under-relaxation Factors in the Solution Controls window control how quickly Fluent tries to come to a solution. In a way, one can consider them analogous to step sizes in other forms of numerical analysis. If the flow is diverging, then these parameters may be too large.
• **Save your current Case and Data.**
• **Open the Solution Controls window.**
• **In the Under-Relaxation Factors portion, change all the values to 0.1. Scroll down to view all variables. There should be a total of seven.**
• **Re-initialize the solution by clicking Solve, Initialize, Initialize.**
• **Select farfield from the dropdown menu.**
• **Click **Init**. In the warning window click **OK** to discard the current values.**
• **Click **Close**.**
• **Start iterating. Set the iteration number to 500.**

The problem with solution variables exceeding set limits will rear its ugly head once more. Keep an eye on the number of cells affected. If this steadily decreases allow the solution to continue. If it increases, cancel the simulation and reduce the Under-Relaxation Factors further. Continue iterating the solution (Note: it may be necessary to re-initialize the flow). Once stable, go back and slightly increase the Factors and restart the solution. Repeat this process every few hundred iterations until the Under-Relaxation Factors are at or close to Fluent’s defaults. The default values are listed below.

<table>
<thead>
<tr>
<th>Fluent Default Under-Relaxation Factors</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Pressure</td>
</tr>
<tr>
<td>Density</td>
</tr>
<tr>
<td>Body Forces</td>
</tr>
<tr>
<td>Momentum</td>
</tr>
<tr>
<td>Modified Turbulent Viscosity</td>
</tr>
<tr>
<td>Turbulent Viscosity</td>
</tr>
<tr>
<td>Energy</td>
</tr>
</tbody>
</table>

Once at the Fluent defaults, iterate using previous methods until force convergence is reached and residuals have decreased several order of magnitude.

Sometimes it may be necessary to reduce the Under-Relaxation Factors by a very large amount. It has been found that reduction of the factors down to 0.1 will stabilize a majority of problems encountered while running cases of external flow. Some, however, may need to be decreased to 0.01. At this point the solutions is just crawling along. One word of caution, before increasing the Under-Relaxation Factors, be sure to save your most recent case file. It is very easy to have a well-running solution only to get over-zealous and increase the Under-Relaxation Factors too quickly causing the solution to crash diverge. This will cause the user to lose all previous efforts. By saving the current case and data the one can import the saved data and address whatever problem caused the divergence. Often it just requires more iteration before increasing the Under-relaxation Factors.

• **Click **File, Write, Case and Data.**
• **Type the desired name for the file and press enter.**
Do not attempt to judge solution convergence unless the Under-Relaxation factors are at or close to the default values. Because they affect the rate of convergence, low Under-relaxation Factors will reduce the slope of the force plots. Upon increasing the Factors, the user should see a jump or discontinuity in the forces and residuals (see figure below). The image below shows a residual plot with jumps occurring after each increase in the Under-Relaxation Factors. This is the solution speeding up towards the “correct” value.

Continuing for a total of 1800 iterations produces a converged drag coefficient of $C_d = 1.547$. This is essentially identical to that achieved earlier. While it needed more iterations than the previous method, adjusting Under-Relaxation Factors can often be the only reliable method of reaching convergence.

**ADDITIONAL EXERCISES**

Listed below are further conditions the reader can simulate to explore supersonic flow around a circular cylinder and controlling unstable solutions.

- **Shock Detachment Distance:** Explore further the relation that shock detachment has with Mach number. Look at how the appearance of the shock changes with Mach number. What does this say about shock strength? Does the separation point on the circumference of the cylinder change with Mach number? Lastly, is there a shock visible in the flowfield at Mach one?
- **Normal Shock Relations:** Using the above results, explore the ratios of the upstream to the downstream flow properties? How well do they agree with theoretical normal shock relations?
- **Grid Dependence:** Attempt to solve the Mach 1.3 flowfield using the higher resolution grid. Compare the two grids to see what differences are present. How many cells does the finer grid contain? How does grid resolution affect solution stability?