2006-709: A WEB-BASED SOLVER FOR COMPRESSIBLE FLOW CALCULATIONS

Harish Eletem, Lamar University
HARISH ELETEM was a graduate student in the Department of Mechanical Engineering at Lamar University. He received his M.S. degree in Mechanical Engineering from Lamar University in 2005.

Fred Young, Lamar University
FRED YOUNG is a professor in the Department of Mechanical Engineering at Lamar University. He received his Ph.D. degree in Mechanical Engineering from Southern Methodist University. He has published many technical papers and presented several papers at international conferences.

Kendrick Aung, Lamar University
KENDRICK AUNG is an associate professor in the Department of Mechanical Engineering at Lamar University. He received his Ph.D. degree in Aerospace Engineering from University of Michigan in 1996. He is an active member of ASEE, ASME, AIAA and Combustion Institute. He has published over 50 technical papers and presented several papers at national and international conferences.

A Web-based Solver for
Compressible Flow Calculations

Abstract

Compressible flow is an important subject in aerospace and mechanical engineering disciplines. This paper describes a web-base solver for carrying out compressible flow calculations. The main objective of the solver is to provide students with a software tool than can be used in the compressible flow course offered in the Department of Mechanical Engineering at Lamar University. The solver has a graphical user interface (GUI) for ease of use and interactivity. The solver is capable of solving typical compressible flows such as isentropic flows, Rayleigh and Fanno flows, normal and oblique shock flows, and Prandtl-Myer expansion waves. The solver provides user with a host of input options to choose from: Mach number, area ratio, pressure ratio, etc. The output of the solver includes the text results as well as graphical plots.

Introduction

Fluid dynamics is a core subject for Mechanical, Aerospace, Civil, and Chemical engineering disciplines. Among many branches of fluid dynamics, compressible flow is an important subject for aerospace and mechanical engineering disciplines as it involves solving high speed air flows in situations such as flight of an airplane or aerodynamic drag of an automobile. In these flow situations, compressibility of the fluid must be taken into account as the density of the fluid cannot be considered a constant. Compressibility of fluid is very important in engineering applications involving flow through nozzles, diffusers, and flows in a scramjet engine.

One of the main obstacles in teaching fluid dynamics to undergraduate students is the lack of visualization and computational tools that enhance and improve learning process of the students. With the widespread availability of multi-media software and hardware tools, development and integration of 2- and 3-dimensional visualization tools and computational tools to the undergraduate fluid curriculum becomes necessary. Therefore, in order to improve the learning process of students, a variety of learning tools such as interactive and multimedia texts, multimedia videos and CD-ROMs, computational and experimental tools, have been developed and used by many educators teaching fluid and thermodynamics courses[1-5]. The development of Internet – specifically the World Wide Web (WWW) has led to unprecedented growth over the last decade in access to information. It offers many advantages: ease of use, quick access, low cost, available without the limitation of time or location, computer platform independent, and flexible in allowing students to control their learning pace. With the advent of JAVA programming language, which offers attractions like platform independence, and development of a single code for many different types of computers, engineering applications can be easily developed for the web.

Motivation and Need for the Solver

For engineering applications involving compressible flow analysis, it is inevitable that the engineer must consult the tables and charts in order to determine the compressible flow properties. However, there are restrictions when using those tables and charts. For example, each table or chart is constructed for a specific value of γ, the ratio of the specific heats for a particular fluid. For other values, tables may not be available and the original equations for a particular flow may need to be solved numerically in order to obtain the desired properties. Moreover, when a situation involving shock waves is encountered, such as an oblique shock or a conical shock, it is very difficult for the user to accurately read the properties from the charts if the desired Mach number is not shown and visual interpolation has to be used. In these situations it is very easy to make mistakes in the calculations. Therefore, in order to resolve the problems mentioned above, many computational tools have been developed and used by many engineers and educators. Regarding compressible flows, some calculators available on the web are VUCALC[6], Compressible Aerodynamics Calculator by William Davenport of Virginia Tech[7], and compressible flow calculator applets developed by Purdue University[8]. VuCalc is based on a program of the same name written by Tom Benson of NASA Glenn as an aid to making calculations in compressible fluid flow[6]. VuCalc can calculate parameters for only five types of compressible flows: isentropic, normal shock, oblique shock, Rayleigh, and Fanno, whereas the present solver can deal with nine different types of compressible flows. Compressible Aerodynamics Calculator can solve for only three types of flows: isentropic, normal shock and oblique shock. Compressible flow calculator applets from Purdue University can calculate parameters for isentropic, normal shock, Fanno and Rayleigh flows with a limited set of features.

From the above discussions, it could be concluded that the compressible flow calculators presently available allow calculations for only a few types of compressible flows with a limited set of features. In addition, none of them include the graphing capabilities as well as generating table values for a given range of a single parameter. The present project therefore is an effort to develop a much more comprehensive Compressible Fluid Flow Solver (CFFS) intended for classroom and educational use.

Objectives of the paper

In the Department of Mechanical Engineering at Lamar University, compressible flow course is an elective course typically offered in the spring semester of each year. It is a three-credit hour class with three 1-hour lectures per week. This paper describes the development of a web-base solver for compressible flows using Java programming language. The main goal of the solver is to provide students with a software tool that can be used in the compressible flow course. The main objectives of the project are

(a) To design and develop a compressible flow solver
(b) To test and evaluate the solver in a classroom environment and
(c) To freely distribute the solver to the educational community for educational uses

Description of the Solver

The solver has a graphical user interface (GUI) for ease of use and interactivity. The programming language used to develop the CFS requires seamless integration with the Internet and the web browser. These requirements are satisfied by the Java programming language developed by the Sun Microsystems. Java programming language was chosen for developing both graphical interface and flow solution procedures, as it is platform independent and the user need only a web browser in order to execute the program. Java has become very popular because of the many features designed to make it operate on the Internet. With this language one can " write once and run anywhere". Developers can write full-fledged applications in Java, whose architecture is much like C and C++. It is freely available from the Sun Microsystems website (http://java.sun.com/). The solver is based on the earlier DOS-version of the solver developed by Professor Fred Young[9, 10] of Mechanical Engineering at Lamar University. The solver is capable of solving many types of compressible flow including isentropic flows, Rayleigh flows, Fanno flows, normal and oblique shock flows, and Prandtl-Myer expansion waves. The solver provides user with a host of input options to choose from: Mach number, area ratio, pressure ratio, etc. The output of the solver includes the text results as well as graphical plots.

Unique Features of the Solver

The current solver, termed Compressible Fluid Flow Calculator (CFFC), has many advantages over existing solvers. Some of the unique features of the current solver are listed below.

(a) It is a comprehensive solver that can solve nine different types of compressible flows such as mass injection and generalized flows
(b) It allows simple calculation and storage of various parameters involved in different types of compressible flows
(c) It can generate the compressible flow tables by allowing users to input the range of the parameter rather than a single value
(d) It allows the user to define the problem parameters such as gas constant and specific heat ratio
(e) Its interactivity and GUI Interface enhances the learning process for students and helps facilitate the educators
(f) The user can choose a gas from more than 40 gases provided in the solver.
(g) The user can also use a gas mixture as a working fluid.
(h) It provides a graphical output that helps understand the variation of each parameter with Mach number
(i) Its graph plotter provides ability to plot graphs for any given range of Mach number (M) and explain f(M) for any given flow
(j) For a particular flow, the CFFC graph plotter provides the approximate value of f(M) for any given Mach number (M)

These features are designed to meet the pedagogical requirements of the compressible flow course as the course typically covers more topics than a typical compressible flow course. For example, the topics of mass injection and generalized 1-D flows are covered in the course at Lamar but these flows cannot be found in typical textbooks dealing with compressible flows. These are specifically included in the present solver as none of the existing solvers provide these types of compressible flow calculations. In addition, the interactivity and graphical visualization provided by the present solver significantly enhances the learning process of students. The visual representation of variation of many parameters with respect to a single key variable, for example

Mach number, in a graphical representation gives students more insight than a table full of numbers that provides the same information.

Based on the course contents and pedagogical requirements, the basic task of CFFC is to

1. Take the input
2. Perform necessary operations and
3. Display the result.

The CFFC is intended to provide a faster, more convenient alternative to printed tables and charts to look up the properties of a compressible flow. It calculates the basic aerodynamic properties for a perfect gas for the given input data from a user. It can handle flow properties corresponding to the nine types of flows:

1. Isentropic Flow
2. Normal Shock
3. Fanno Line
4. Rayleigh Line
5. Isothermal Flow
6. Mass injection
7. Generalized 1-D
8. Prandtl-Meyer Expansion Wave
9. Wedge Oblique Wave

In a typical software package, the front end takes care of the user input using a graphical user interface (GUI). The back end implements the solution algorithm and solves the problem that the user asks by using the front end of the software. For the current solver, the **Front End** of CFS takes the data from the user and displays the input data while the **Back End** performs the operations (or calculations) in the background.

Front End

The Front End of CFS is developed using Java Swing graphical package from Sun Microsystems. It consists of an applet, frames, dialog boxes, file choosers, etc. The screen shot where the user chooses the type of calculator is shown in Fig. 1.

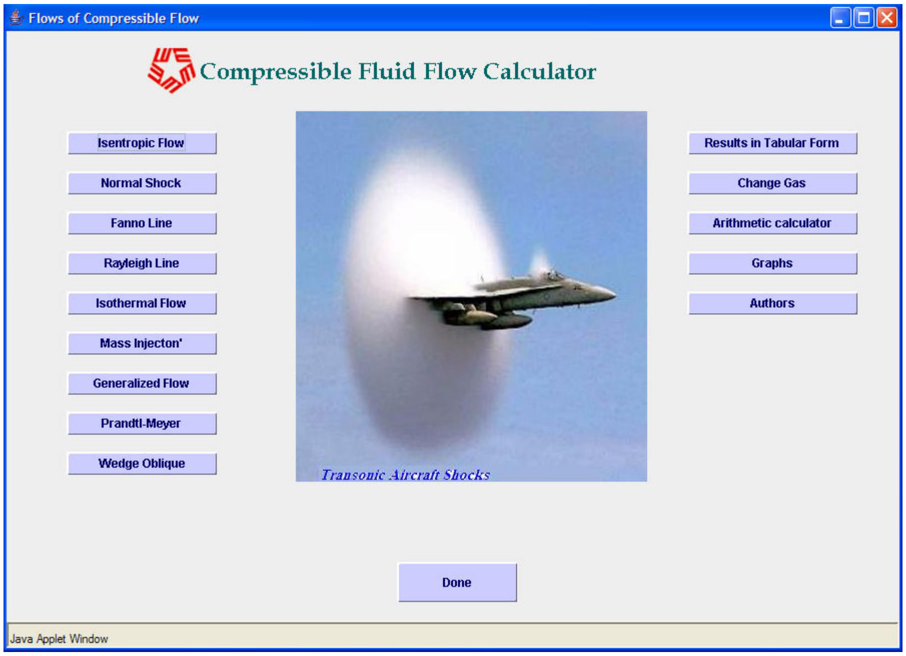


Figure 1 The Opening Screen of the CFFC

Using the screen shown above, the user can select the type of flow to be solved. After one type of flow is chosen, the use needs to click the next button to activate the corresponding calculator. For each flow, the user has the option to change the system of units, or the type of gas or gas mixtures, and a calculator to compute or modify the results from the calculator. The units used in the calculator can be changed to either SI System or English Engineering System or English Gravitational System as shown in Fig. 2.

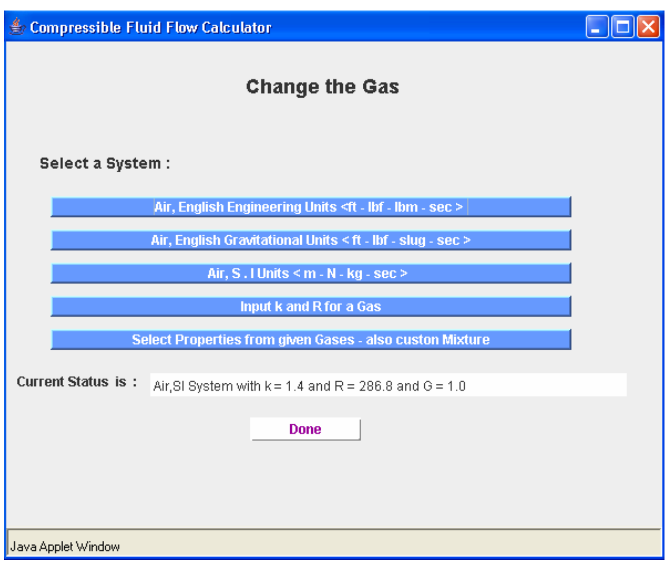


Figure 2 The Program Screen to Change the Parameters of the Problem

The CFS provides over 40 gases to choose from, among them many common gases such as air, helium, and argon. The screen for the user to specify the gas is shown in Fig. 3 below.

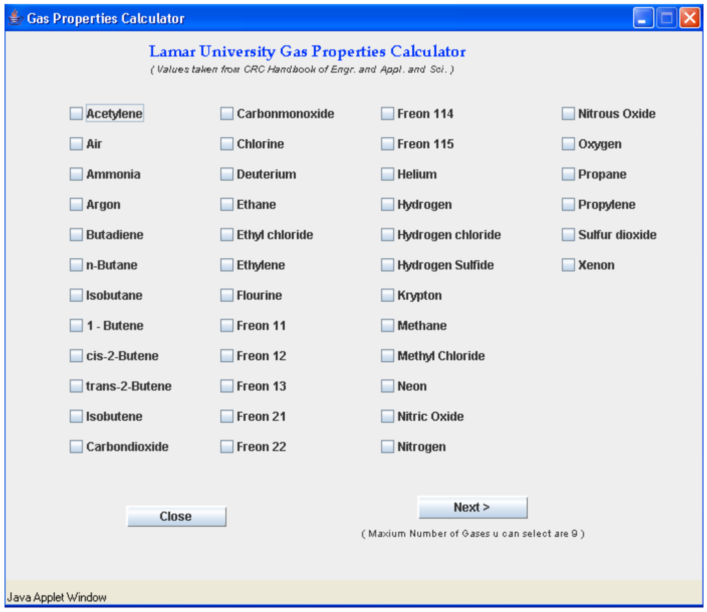


Figure 3 The Program Screen to Choose the Gas

In addition, it has the option to use a gas mixture of the given gases or the user- defined mixture of gases. If the user chooses to specify a new gas, he needs to input the relevant properties of the gas such as gas constant and specific heat ratio in the screen shown in Fig. 4.

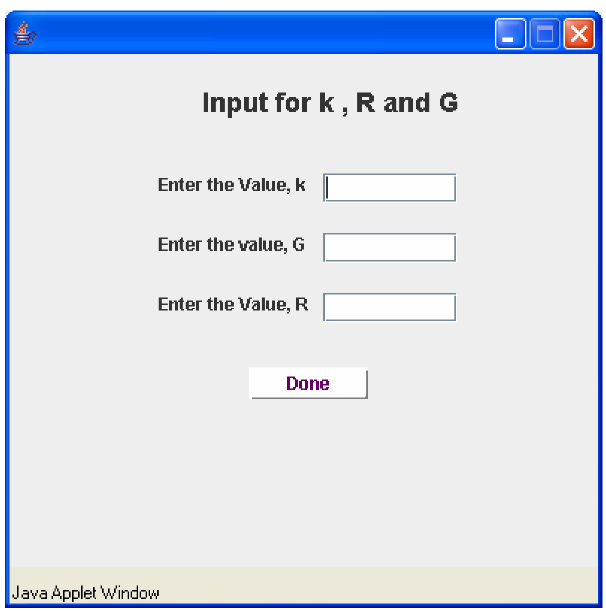


Figure 4 The Program Screen to Specify the Properties of the Working Fluid

In a typical compressible flow problem, the Mach number or one of the critical parameter such as area ratio is generally known. Using that known parameter, all other relevant parameters of the problem can be obtained explicitly. For example, in an isentropic flow, it can calculate any of the other following properties if one of the following properties is given:

1. Mach number.
2. Area Ratio.
3. Temperature ratio.
4. Pressure ratio.
5. Density ratio.
6. Critical Mach number.
7. Static Mass flow ratio.
8. Stagnant Mass flow ratio.

To calculate the density ratio for a given area ratio, go to calculator, click on Isentropic Flow, click next and then click on the known property. In this case, click on the choice of area ratio and enter the correct value when prompted for input. Then click on compute, and the results are displayed. The screen shot of an isentropic calculator is shown in Fig. 5.

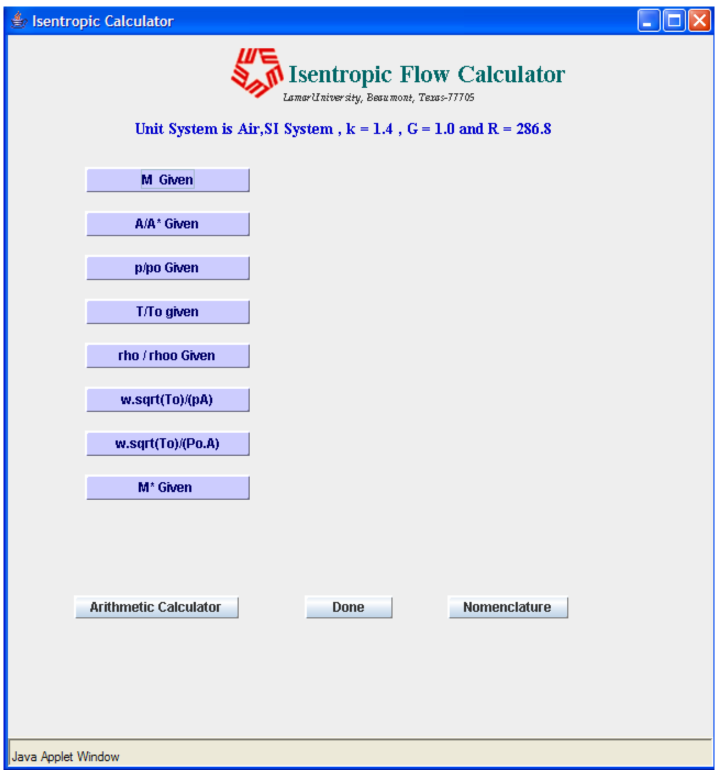


Figure 5 The Program Input Screen of the Isentropic Flow Calculator

Figure 6 is the output screen of the isentropic calculator. Note that there are two solutions (one for subsonic flow regime and another for supersonic flow regime) related to the input parameter and both are shown in Fig. 6.

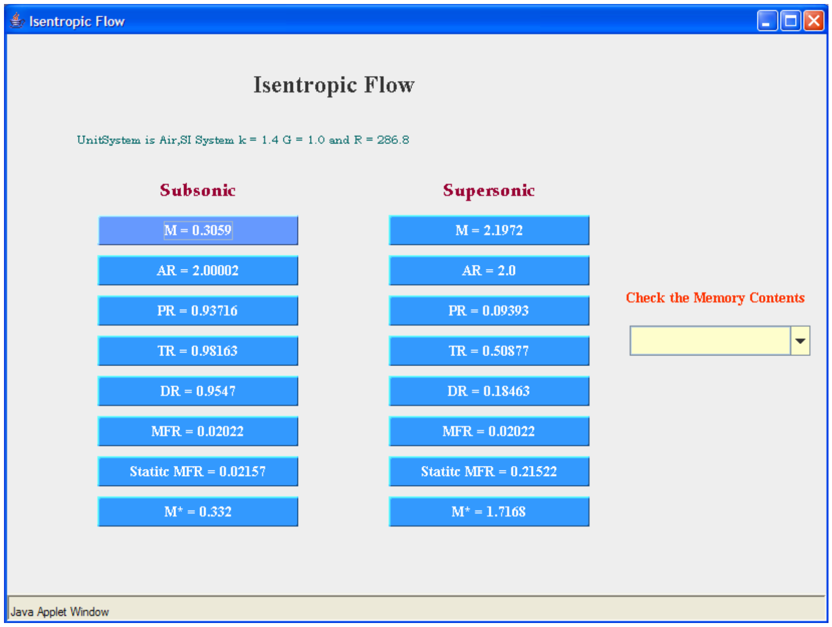
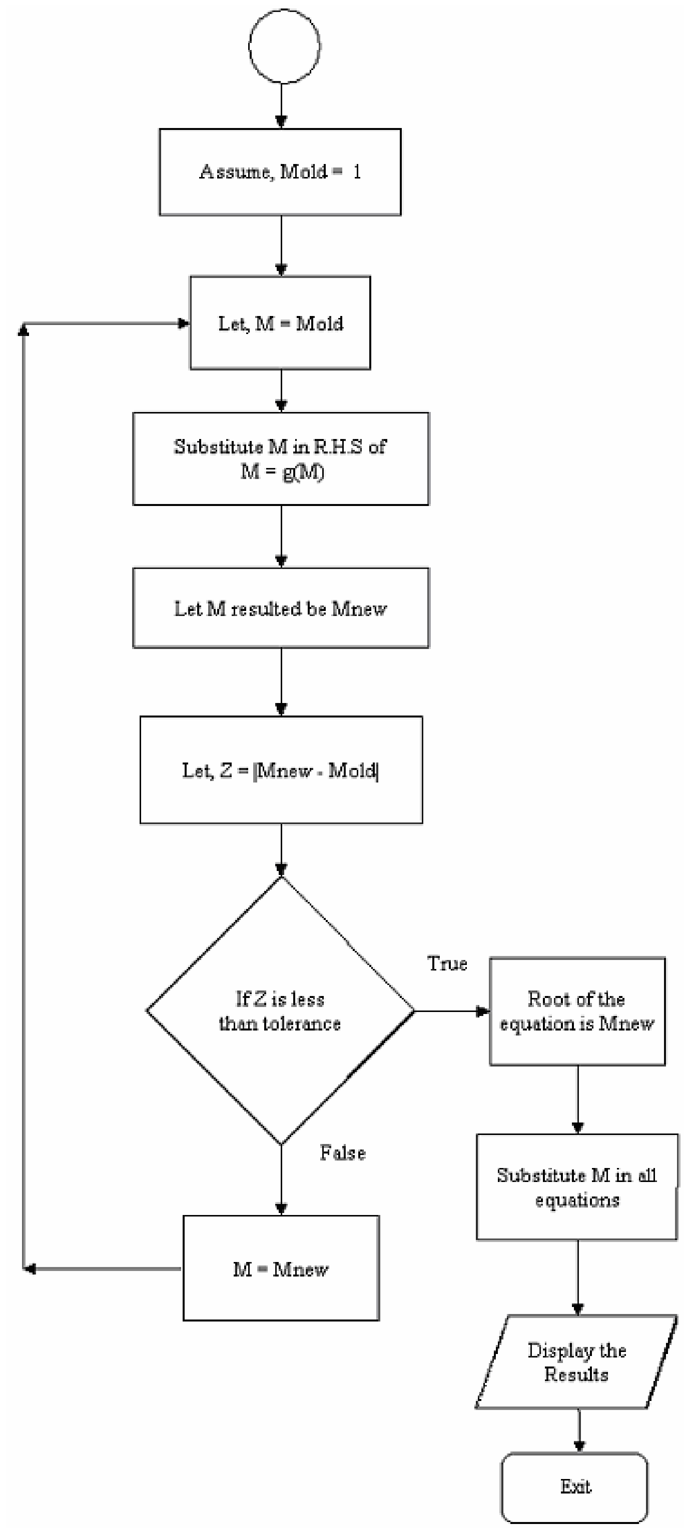


Figure 6 Output Screen of the Isentropic Calculator

The present solver can solve any type of compressible problem including problems where iterative solution procedure is required. For example, in shock flow calculation the knowledge of the upstream Mach number can provide all other parameters for the problem by direct substitution into the analytical expressions of the solution. However, if any other parameter is chosen, all the other parameters cannot be calculated by direct substitution into the analytical expressions for the solution. Then, the solver will resort to the iterative solution method as described next. In other words, all the above equations are of the form: $Y = f(M)$, where Y and all other equations can be obtained by direct substitution of Mach number M. On the other hand, if Y is known, the following steps are taken to calculate the Mach number M.

1. Substitute Y into the above equation which yields to $M = g(M)$.
2. Assume $M_{old} = 1$
3. Let $M = M_{old}$
4. Substitute M in the R.H.S of the above equation, $M = g(M)$.
5. Let the new Mach number be M_{new}
6. Let Z be the absolute difference of M_{old} and M_{new}.
7. If Z is less than allowed tolerance, then M_{new} is the root of the equation .Otherwise, $M = M_{new}$ and go to step number 4.
8. Substitute M in all the equations to get different parameters of isentropic calculator.

The algorithm of the solver can be summarized in the form of flow chart as shown in Fig. 7.



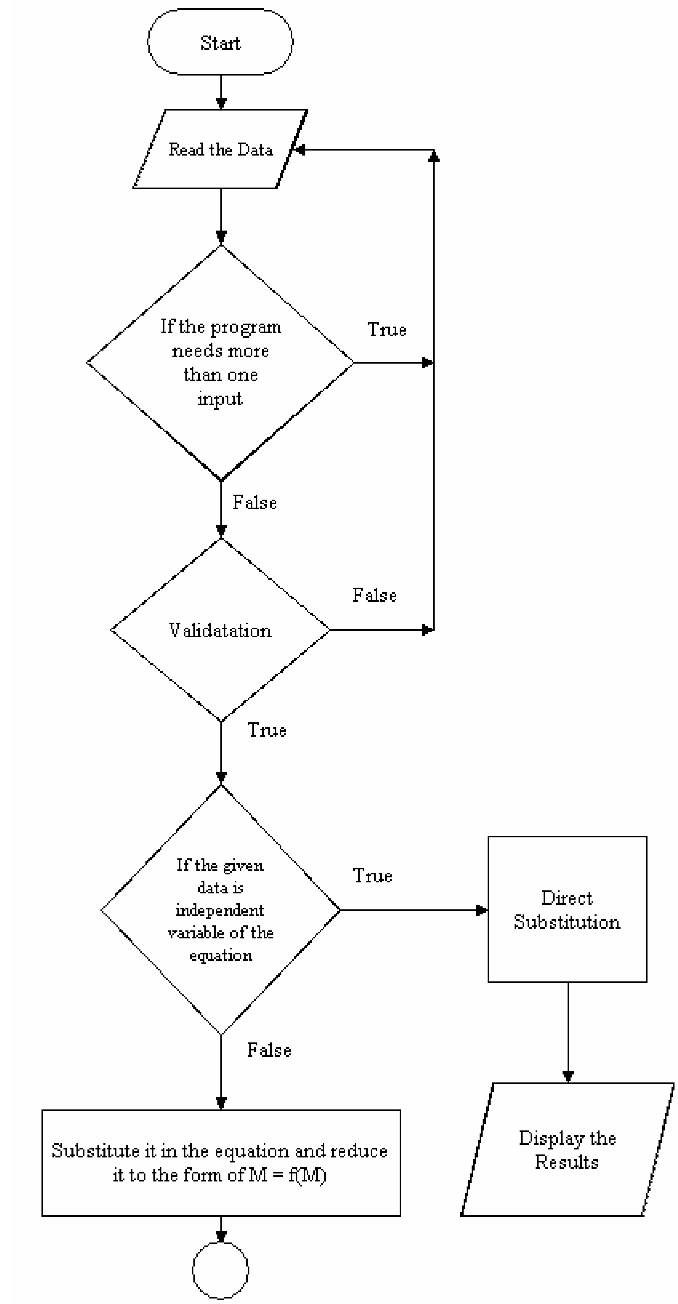


Figure 7 Flow Chart of the Implicit Algorithm

The results from a series of calculations for Rayleigh flow and generalized flow are plotted together with the analytical results in Figs. 8 and 9 to validate the performance of the CFFC. As can be seen from these figures, the CFFC gives results that are almost identical to the analytical results verifying the robustness of CFFC for different types of compressible flow.

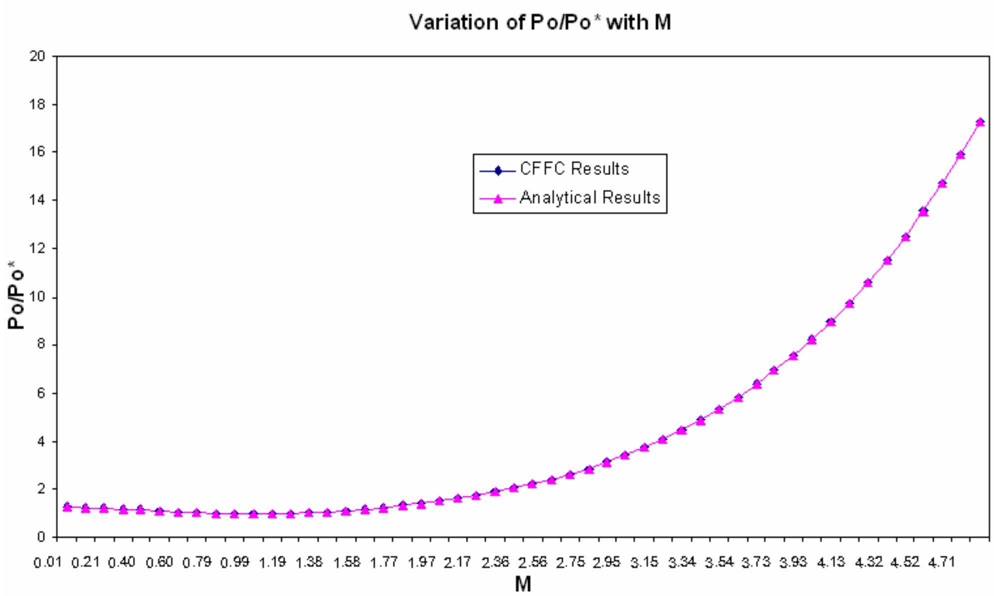


Figure 8 Comparison of CFS Results and Analytical Results for a Rayleigh Flow

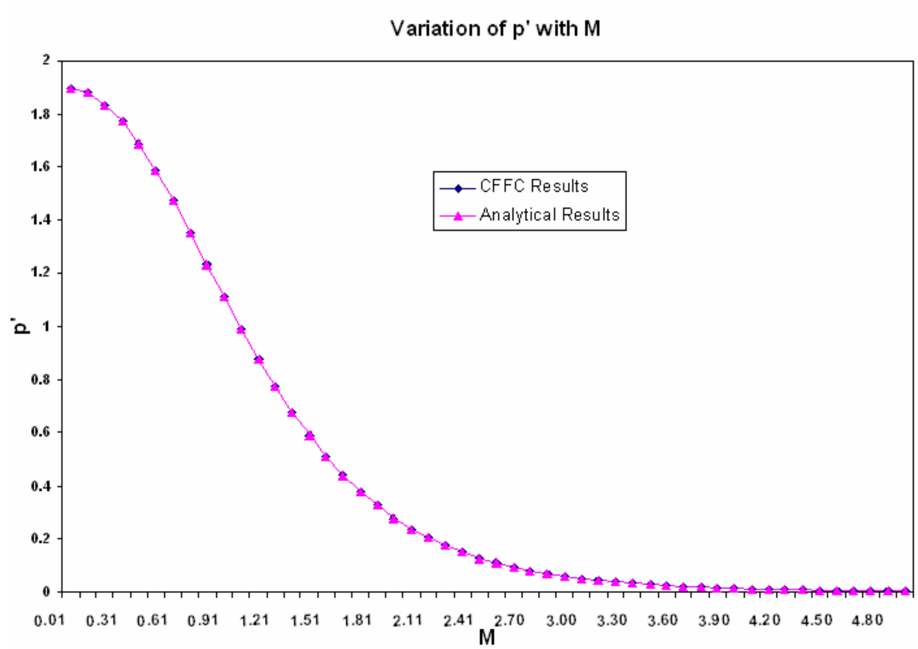


Figure 9 Comparison of CFS Results and Analytical Results for a Generalized Flow

Graphical Results

As discussed before, one of the unique features of CFFC is its ability to plot graphs and obtain results from the graph. Figures 10 and 11 are two graphs drawn by CFFS graph plotter for an isentropic flow and a Fanno flow respectively. These graphs were implemented using Java 2D graphing package by Brookshaw[11].

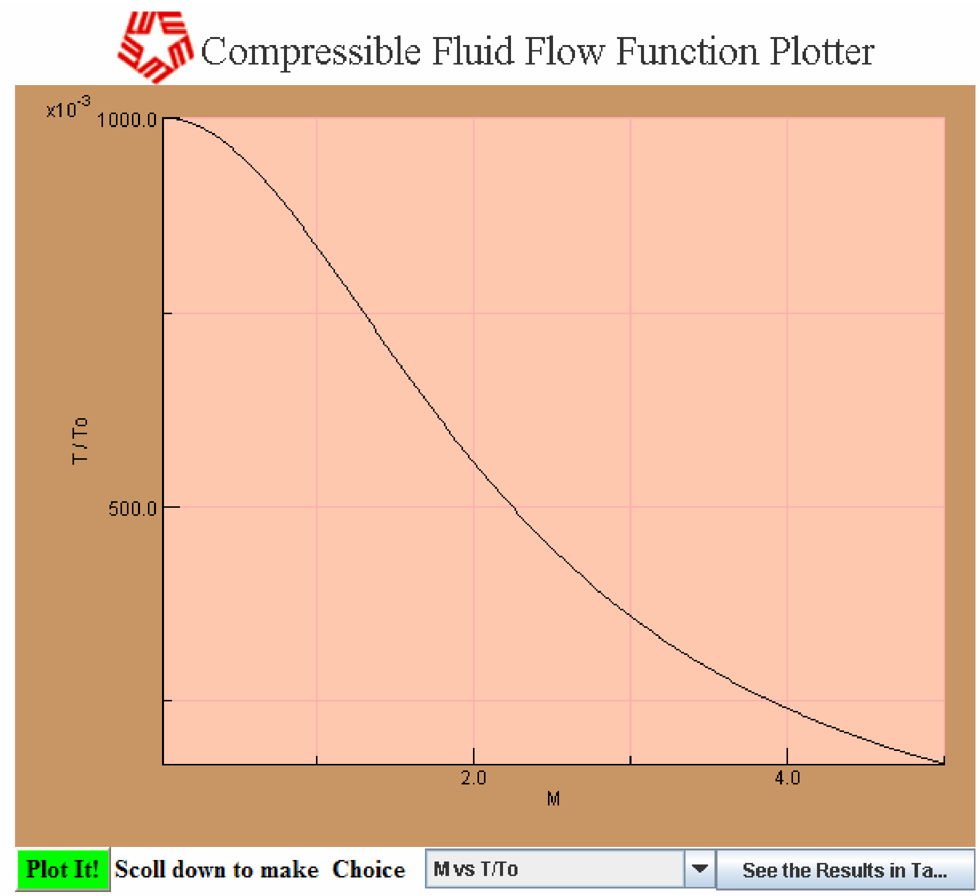


Figure 10 Plot of T/T_0 vs Ma for an isentropic flow

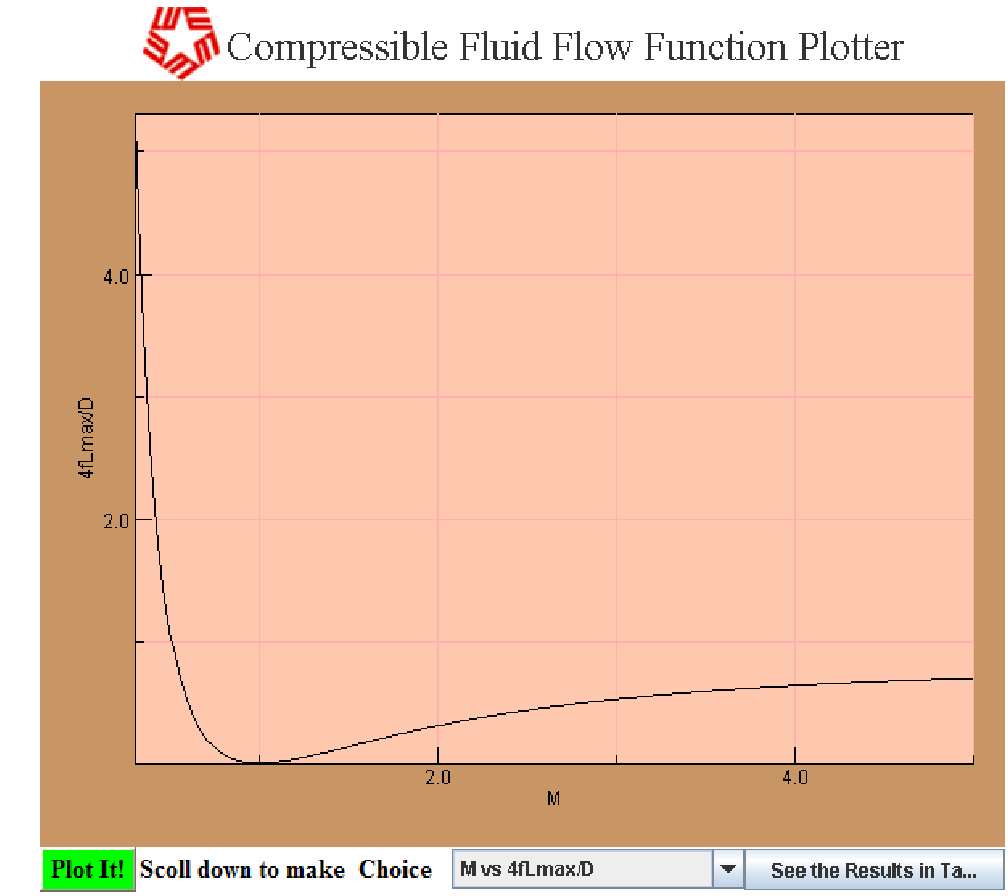


Figure 11 Plot of Friction vs Mach Number for a Fanno Flow

Distribution of the Solver

The solver was developed with the intention of free distribution to the educational community and other interested users. Thus, the solver will be made available for free download from the web site of the Mechanical Engineering department. At present, anyone who wants to use the CFFC can send an e-mail to the author at aungkt@hal.lamar.edu and request the program.

Feedback and Improvements

The solver development was completed in June 2005. The compressible flow course is offered only in the spring semester. As a result, it is being used in class right now, and the responses and feedback from the students will be made available after the semester is over. Future improvements on CFCC include addition of Conical Shock, Flow Simulator and Supersonic airfoil and Area graphs for Oblique shock waves and Prandtl-Meyer flow.

Acknowledgements

The authors would like to acknowledge Leigh Brookshaw for his 2D Java Graphing package without which the graphing capability of CFFC will not be possible.

Bibliography

1. *CALF*. (1996). Retrieved July 2, 2003, from University of Glasgow and University of Paisley, United Kingdom, Web site:http://cvu.strath.ac.uk/courseware/calf/CALF/index/web_calf.html.
2. Ham, F.E., Militzer, J., & Bemfica, A. (1998). *CFDnet: Computational fluid dynamics on the internet*. Retrieved November 12, 2002, from http://www.cfdnet.com.
3. Bhattacharjee, S. 2002. *TEST (The expert system for thermodynamics)*. http://kahuna.sdsu.edu/testcenter/.
4. Appanaboyina, S. and Aung, K., (2004), "Development of a VRML Application for Teaching Fluid Mechanics," Proceedings of 2004 ASEE Annual Conference, Nashville, Tennessee.
5. *The Java virtual wind tunnel*. (1996). Retrieved July 2,2003, from Massachusetts Institute of Technology, Cambridge, Web site: http://raphael.mit.edu/Java/.
6. Carmichael, R. 2005. *Public domain aeronautical software*. http://www.pdas.com/vucalc.htm
7. Compressible Aerodynamic Calculator. http://www.engapplets.vt.edu/fluids/compresssibleAero/ifmCompAero.html
8. Purdue University. *Compressible flow calculators.* http://widget.ecn.purdue.edu/~meapplet/java/comp_calculator/Index.html
9. Young, F.M. 1993. *Compressible flow calculator (software).*
10. Young, F., (1996), "The Use of MathCad as a Lecture Aid for Compressible Flow," Proceedings of 1996 ASEE Annual Conference and Exposition.
11. Brookshaw, L. 1998. *Java 2D graph package.* http://www.sci.usq.edu.au/staff/leighb/graph/