NASA Technical Memorandum 106587

IN 61
5514
18 P

On the Use of Computers for Teaching Fluid Mechanics

Thomas J. Benson
Lewis Research Center
Cleveland, Ohio

**National Aeronautics and
Space Administration**

(NASA-TM-106587) ON THE USE OF
COMPUTERS FOR TEACHING FLUID
MECHANICS (NASA. Lewis Research
Center) 18 p

N94-32773

Unclas

G3/61 0005514

On the Use of Computers for Teaching Fluid Mechanics

Thomas J. Benson*
Internal Fluid Mechanics Division
NASA Lewis Research Center
Cleveland, Ohio 44135

April 8, 1994

ABSTRACT

Several approaches for improving the teaching of basic fluid mechanics using computers are presented. There are two objectives to these approaches; to increase the involvement of the student in the learning process, and to present information to the student in a variety of forms. Items discussed include: the preparation of educational videos using the results of computational fluid dynamics (CFD) calculations, the analysis of CFD flow solutions using workstation based post-processing graphics packages, and the development of workstation or personal computer based simulators which behave like desk top wind tunnels. Examples of these approaches are presented along with observations from working with undergraduate co-ops. Possible problems in the implementation of these approaches as well as solutions to these problems are also discussed.

INTRODUCTION

Recent advances in computer related technologies could revolutionize the teaching of undergraduate fluid mechanics in the twenty first century. The world wide networking of computers is ushering in a new age of information and communication. The introduction of CD-ROM storage devices have enabled entire encyclopedias to be stored and accessed from a small disc. The availability, computing power, and memory capacity of personal computers exceeds the mainframes of the past. Advances in mainframe size and speed, computing algorithms and software, and parallel processing have enabled the computational

analysis of ever more complex problems. The coupling of this hardware and software growth with new user-friendly operating systems, windowing capability, and the development of graphical user interfaces (GUI) has created an entirely new computing environment. Interestingly, it is an environment in which undergraduate students are quite comfortable, having grown up with video games and personal computers at school or in the home. The challenge for educators is to learn, develop, and apply the new technology to the traditional problems of fundamental understanding in fluid mechanics.

The teaching of undergraduate fluid mechanics has always been a two phase process; the development of intellectual or mental knowledge, and the development of intuitive or gut knowledge. Intellectual knowledge would include the recognition of terminology, concepts, basic assumptions, and the derivation of the appropriate flow equations. Intuitive knowledge would include the recognition of the relationships between variables and the understanding of how things worked. Lecture classes most often provides intellectual knowledge while laboratory classes provide intuitive knowledge. When the author was an undergraduate, one developed an intellectual understanding of boundary layer theory using Schlichting (1968), and developed an intuitive understanding using the movies by Shapiro (1961). Computational tools to assist in the development of intellectual knowledge would include symbol manipulation packages, such as MACSYMA (Rand(1984)), which could aid in the derivation of equations. One could use the storage, retrieval, display, and networking of computers to make basic information more quickly and readily available to students. There are, for example, software packages now available in the

medical profession which allow the student to view anatomical pictures quickly and interactively. Similar types of software could be developed for fluid mechanics which would store and display existing texts.

It is, however, in the development of intuitive knowledge that computers could play a major role, particularly at a time when laboratories are becoming even more expensive to operate and maintain. Personal computers and workstations are particularly well suited for the development of intuitive knowledge. They are readily available, relatively inexpensive, and becoming much easier to use. The output from a modern personal computer is principally visual; the most perceptive sense used in education. Some more advanced machines include audio, which can reinforce the lesson. The proper software can make computers highly interactive, causing the student to respond in some fashion to the lesson and not merely observe it. With motion available on the computer screen, the medium is truly three dimensional; two space dimensions plus time, or the illusion of three space dimensions. Traditional books, paper and chalkboards are only a two dimensional medium. The added dimension available in the computer screen can dramatically change the student's perception and understanding of the fluids problem.

Computers can assist the teaching of fluid mechanics in several ways. The output from large CFD codes are often displayed at conferences through video tapes. Videos which highlight a fundamental fluids problem can now be used in the normal classroom environment. For fundamental flow problems, dedicated computations and videos can be produced solely for educational purposes. Videos are produced using the output from a CFD code as input to a graphics package running on a workstation. It is possible for the CFD output data sets and the graphics packages to be made available to universities for student use. With the appropriate scripts and some experience, the student can use the graphics package to explore the flow solution and develop the student's own understanding of the flow phenomenon. Considering the computing power of some personal computers and workstations it is now possible to produce software packages which can both solve and graphically display basic fluids problems on the workstation. Several personal computer based software packages have been developed recently (Koening and Hodge, 1993; Fox and McDonald, 1992; and Mat-

tingly, et.al., 1987) to assist in the teaching of fluid mechanics and propulsion. This paper will present some ideas and examples of two packages by Benson (1994a and 1994b) which are under development at the NASA Lewis Research Center for the teaching of fluid mechanics to undergraduate and graduate students. Eventually the large CFD codes can also be made available for students to formulate solutions to flow problems.

A particularly intriguing approach to using computers in education is the development of flow simulators which act like desk top wind tunnels. On a workstation, the student is presented with a physics problem in one view window. In another window is a control panel which the student uses to vary the conditions of the physics problem. In a third window raw output data from the problem is presented to the student. The student can then select which data to analyze, and in another window record the data and plot the data on a view screen. If the problem is properly chosen, the equations governing the flow can be almost instantly solved on the workstation and the entire package behaves like a wind tunnel simulator. A preliminary flow simulator has been developed by Benson (1994a) for the basic problem of supersonic flow past wedges. It could be rather easily modified to consider alternate physics problems, such as a supersonic nozzle, a sting-mounted wedge at angle of attack, or a diamond airfoil, The basics of the code could be retained and other classes of fluids problems coded, including potential flow, conformal mapping, shock tubes, stability and control, and basic airfoil theory. Expanding the idea to other areas of science, basic thermodynamics, magnetics, electrostatics and optics problems also easily lend themselves to presentation in this way.

EDUCATIONAL VIDEOS

Educational videos can be used to supplement regular classroom lessons on various flow phenomena if several problems are overcome. Since the advent of the supercomputers in the early 1980's many highly detailed three dimensional flow calculations have been performed and the results presented at technical meetings by means of video tape. There are, however, several problems associated with these videos. The first problem is identifying what videos exist for particular flow solutions since a video presentation is seldom noted in conference proceedings

or schedules. This problem could be overcome by using the Internet system to build a catalog of existing videos. Inquiries could be posted in various newsgroups and the results catalogued for future reference. The second problem is that flow solutions presented are often too complicated for undergraduates. For instance a video describing flow around the forebody of a fighter aircraft contains too many phenomena (vortex shedding, boundary layer buildup, shock waves, separated flows, ...) for detailed understanding by undergraduates. Single phenomenon calculations are preferred and this may require that new calculations of selected problems be performed. The selection of the appropriate problems for study must be provided by the educational community. The third problem is the variation in style and content of existing videos. For instance, some videos may be working level tapes; without captions, problem descriptions, voice over, or any other kind of explanation, while others might be described as public relations tapes with lots of music and visual impact but little technical value. A possible solution to this problem would be the preparation of a series of videos by some centralized group with the videos prepared under guidelines provided by the educational community. Old videos can be reworked in a standardized style.

To demonstrate some of the possibilities of using CFD to prepare videos for educational purposes, a video has been prepared which shows in great detail the periodic vortex shedding from a circular cylinder, the Karman vortex street. The calculation of the flow field was performed by Kim and Benson (1992) as part of a code development and validation study. The flow is incompressible, at Reynolds number of 100 based on free stream velocity and cylinder diameter and the calculated Strouhal number is .16. The video was prepared at the NASA Lewis Graphics and Visualization Laboratory using FAST, the Flow Analysis Software Toolkit, (Walatka *et.al.* (1994)). The video runs about 30 minutes, without music or voice-over, but with several still shots separating the action and describing the flow variables and scenes which follow. The video presents the same flow field in several different ways; velocity vectors, streamlines, particle traces, color-tagged particles, vorticity contours, and injected fluids. This flow field is time dependent and the video allows the student to visualize the flow field changing with time; the figures

in this report cannot even begin to present this information. Color is used in the videos to highlight various features and this aspect is also not available in this paper. Each of the different representations present the viewer with information not available in the other representations. For instance, the particle traces shown in Figure 1 demonstrate the formation of the top and bottom alternately shed vortices. In the video, the mixing of the flow at the rear of the cylinder is quite apparent. Figure 2 shows the same flow field at the same time but now contour plots of vorticity are shown. The student can see the high shear regions on the front of the cylinder and the fact that the vorticity is quickly dissipated downstream, facts not available in the particle traces of Figure 1. In Figure 3, fluid is injected from inside the cylinder and convected downstream. This technique can be compared to experiments, but again presents a different view of what occurs in the flow field compared to Figure 1 or Figure 2. Combining all of the different representations, the student can come to a better intuitive understanding of this physical process than is available from any one presentation, or from a textbook picture.

COMPUTER GRAPHICS

Computer graphics have been used for many years to display the output from large CFD codes. The trend in recent years has been towards workstation based highly interactive packages. These packages post-process data sets which are created by the flow solver and contain the results of the flow calculation. Because the two processes of analyzing the results and producing the results have been split, a student can learn the basic physics of certain flow problems without first learning all of the intricacies of CFD. The flow results can be computed by someone else. All that is required for this type of educational experience is a workstation, a graphics package, the computed data sets, and some experience with the software and hardware. As previously noted, workstations are becoming much more available and inexpensive, and many universities now have computing labs equipped with workstations. The new graphics packages, such as FAST, come equipped with on-line demos, scripts, and some flow data sets. Experience with co-op students at NASA Lewis has shown that undergraduates can quickly learn these new packages

and usually show great interest and initiative in using the packages.

As with the educational videos, there are some problems to be overcome before computer graphics can be widely used in undergraduate education. Most computer graphics packages are licensed software which must be purchased from vendors and are usually expensive. Perhaps the use of these packages for educational, not commercial, purposes and the promise that a whole generation of fluid dynamicists would be schooled in the use of some package could lower the expense. Finding the data sets to analyze is a problem which, like the videos, could be solved either through an intense literature search or through an Internet broadcast. The use of three dimensional flow fields is particularly encouraged in this effort although the storage requirements can be quite high. Like the videos, it may be desirable to request certain fundamental flow calculations to be completed solely for the purpose of producing data sets for analysis. Once the appropriate data sets are located or generated, it may be desirable to store and maintain them at some central location, either a university or a government workstation accessible via the Internet. It may also be desirable to create some beginner's scripts for the graphics package to maximize the efficiency of the user's time. However, the central focus of using computer graphics should always be to maximize the student involvement in the analysis. The interaction of the student and the flow problem will produce the desired result.

As an example of the types of data sets and flow problems which have been analyzed by undergraduate co-ops at NASA Lewis, Figures 4 and 5 show Mach 3.0 viscous flow past a sharp fin on a flat plate as computed by Anderson and Benson (1983). Figures 6 and 7 show subsonic viscous flow past a jet in cross-flow as computed by Kim and Benson (1993). The graphics package used in both analyses is FAST, although undergraduate co-ops have also studied the sharp fin problem using the older PLOT3D package.

The sharp fin case is interesting because it involves a three dimensional shock wave boundary layer interaction. Figure 4 shows a black and white representation of a colored screen dump from a workstation. The figure shows the fin geometry at the top, the flat plate at the bottom and polygon-filled contour plots of Mach number at three computational planes. The co-ops learned to recognize shock waves in the free

stream from the abrupt change in color, boundary layers along the walls from the dark contours near the surface. The co-ops could see the interaction between the shock and boundary layer in the thickening of the boundary layer in the vicinity of the shock, the thinning back towards the wedge, and the curving of the shock in the vicinity of the wall. Figure 5 shows the result of particle traces near the wall which produces several well known features including the line of coalescence, and the downstream conical area. The students create these patterns by invoking the streamline generator and placing a "probe" at some points in the flow field interactively as indicated by the white cross hairs in Figure 5. The student can then view the results from different orientations by using a mouse to translate or rotate the picture.

The jet in cross-flow problem is interesting because the geometry is quite simple and the flow field contains secondary vortices and boundary layer separation and re-attachment. Figure 6 shows two views of the secondary vortices generated by the interaction of the injected flow with the approaching boundary layer. The top view shows a half symmetry view from the side while the bottom view is from upstream looking across the injection jet. The co-ops often relate this flow pattern to smoke issuing forth from a chimney on a windy day. Figure 7 shows a topological analysis of the flow separation around the jet. The view is down onto the flat plate and the separation and reattachment lines are noted. The co-op who produced this figure was learning topology while studying fluid mechanics and this flow problem and graphics package satisfied both pursuits.

FLOW SIMULATORS

The newest and most promising effort to use computers in the teaching of fluid mechanics involves the use of flow simulators for specific fluids problems. Because of the increased computing power of modern personal computers and workstations it is possible to build packages which solve the flow equations for simplified problems nearly instantaneously. Coupling the solver to a GUI produces a flow simulator. The student can then develop the understanding for the flow problem through interaction with the simulator, much as students formerly developed understanding through laboratory courses. The GUI also allows the results of the flow calculations to be

4

presented to the student in several different forms which assists in the understanding of basic flow phenomenon.

In the past eight months two simulators have been developed: the first solves for supersonic flow past a single wedge, two opposed wedges, or two wedges in series; the second solves for the flow in two dimensional external or mixed compression inlets. The first package is intended for undergraduate education and contains some unique features to encourage the students interaction with the package. The second simulator is intended for graduate education or for preliminary design in industry. In this discussion, we shall concentrate on the first simulator and the reader can find more details of preliminary versions of both simulators in Benson (1994a and 1994b). Some examples of the results of the simulators are given in Figures 8 through 14 which are screen dumps from the simulators.

Figure 8 shows the basic layout of the simulator which is divided into four main sections: the main view window is in the upper left, the plotter view window is in the upper right, the output box is in the lower right, and the input box is in the lower left. The main view window shows the geometry, the shock (or expansion), and labels for the hinge and the wedge. On the workstation, these features are color-coded, but are presented here in black and white. The wedge appears as the lower nearly horizontal line with the small semi-circle denoting the hinge location. The upstream flat portion is tagged "0" while the movable wedge is tagged "1". The shock appears as the upper nearly diagonal line originating at the hinge location and flow is from left to right above the wedge surface. The plotter view window is located to the right of the main view window. The student selects which sets of variables to plot using the input box. The computed output flow conditions are displayed in the output box below the plotter view window. This data is displayed in two ways; numerically in the row of boxes at the left, and as bar charts to the right. Each bar is a different color corresponding to a different flow variable. As the flow conditions are changed in the input box, the recalculated numbers are displayed and the bar charts move much like a thermometer. This type of visual output allows the student to immediately sense in what direction the flow variables change and by how much for a given input. The input box is located to

the left of the output box. It includes some buttons to select a problem for study and four sub-panels to vary conditions in the problem. The contents of the sub-panels depend on the problem chosen for study. In Figure 8 a single wedge problem is indicated by the darkened "light" on the button and only three of the four sub-panels are required for input. The first sub-panel controls geometry and free stream flow conditions. The lower two sub-panels are used to control the plotting of data; two sub-panels are used to allow flexibility in the types of plots one can generate for different physics problems.

When generating plots, the simulator behaves like a desk top wind tunnel. The student chooses the independent and dependent variables for plotting in the sub-panel marked "Variables". As the choices are made, the axes in the plotter view window are automatically labeled and scaled. In Figure 9, the student has chosen to plot static pressure ratio versus wedge angle as indicated by the lights on the buttons. The Mach number then becomes a parameter for generating curves. The student sets a value of Mach number then varies the wedge angle to any desired value and presses the "Take Data" button. At this point a "*" appears on the graph corresponding to the chosen value of independent variable (wedge angle) and calculated value of dependent variable (pressure ratio). The student then uses the input sub-panel to select a new wedge angle and again "Take Data". A new point appears on the plot and the procedure is repeated to a maximum of twenty five data points. The data can be taken in any order, so the student can fill in interesting portions of the curve. In Figure 9, six data points have been taken. When the student has completed a trace, the "End Trace" button is pushed, a solid color coded line is drawn through the data, and a colored label with the value of the parameter is affixed as shown in Figure 10. The student can then choose a new value of the parameter and begin a new trace as before. Figure 10 shows the screen dump after the student has begun a new trace at Mach number equal to 2.0. In this example, ten data points have been taken along the new trace, and the old trace has been labeled in the upper right corner of the plotter window. The student can put up to five traces on the plotter. To begin a new plot, the student pushes the "New Plot" button, the old plots are erased, the count boxes for traces and data are reset to zero, and the "Variables" buttons

can be used to pick new independent and dependent variables.

This simulator can present different results to the student depending on the flow problem. In Figure 11, supersonic flow past opposed wedges has been chosen. The sub-panels are different than before since the student can now vary the angle of the second wedge and the distance between the wedges. The display in the main view window now contains the second wedge labeled near the top of the figure and the multiple flow zones are also tagged. This flow problem includes a slip line between zones 3 and 4 which is noted by the dashed line, and the zone selection button in the output window can now be used to display the values of all the flow variables in any given zone; in this example zone 3 is displayed. Because of the added complexity of this problem, the plotter is used in a different way than it was for the single wedge problem. When the plotter is invoked, as shown in Figure 12, a set of cross-hairs appear in the view window. These cross-hairs can be moved by using the sliders in the window labeled "Probe". The student can then produce plots of flow variable versus position rather than flow variable versus Mach number or wedge angle as in the single wedge problem. For this example, static pressure versus X-distance along the ramp has been selected and data points have been recorded in the plotter. The process of recording the data is the same as in the previous problem.

No educational tool would be complete without an examination; a question and answer box has been added at the lower right corner as shown in Figure 13. The questions and answers are stored in a data set which the simulator accesses. Teachers (or for that matter students!) can edit this data set and add, modify, or delete questions and answers as required. The questions can appear in the window either sequentially or randomly as chosen by the student with the appropriate buttons. The current sequential set of questions and answers have been chosen to promote an interaction of the student with the simulator and to bring out several important topics regarding compressible aerodynamics. To use this feature the student pushes the "Question" button for a question, uses the simulator to obtain an answer, then presses the "Answer" button to check the answer. As currently configured, the simulator runs through the same questions every time the simulator is invoked.

This portion of tool could be modified to present questions from different data sets or even questions with a timer but without the provided answers - a true examination tool.

As an additional example, Figure 14 shows the display employed in the inlet simulator package. This simulator does not provide for the plotter used in the undergraduate package but is more concerned with performance aspects of a high speed inlet, such as its drag and total pressure recovery. The simulator includes a model for the subsonic portion of the inlet duct based on wind tunnel testing. Again this package was chiefly intended for design and graduate education because of the multiple problems solved here.

SUMMARY

High speed computer mainframes, personal computers, and workstations coupled with interactive operating systems and computer graphics provide many opportunities for new techniques to teach basic fluid mechanics. This paper has presented several ways in which the new technologies can be utilized including the preparation of educational videos using the results of CFD calculations, the analysis of flow solutions by students using the results of CFD calculations and post-processing graphics packages, and the development of workstation or personal computer based simulators which behave like desk top wind tunnels. Examples of these approaches have been presented along with discussion of some possible problems in their implementation.

ACKNOWLEDGEMENTS

6

REFERENCES

Anderson, B.H., and Benson, T.J., 1983, "Numerical Solution to the Glancing Sidewall Oblique Shock Wave/Turbulent Boundary Layer Interaction in Three Dimensions," AIAA Paper No 83-0136.

Benson, T.J., 1994a "An Interactive Educational Tool for Compressible Aerodynamics", AIAA Paper 94-3117.

Benson, T.J., 1994b "An Interactive, Design and Educational Tool for Supersonic External-Compression Inlets", AIAA Paper 94-2707.

Fox, R.W., and McDonald, A.T., 1992, *Introduction to Fluid Mechanics*, John Wiley, New York, 4th ed.

Kim, S.-W. and Benson, T.J., 1992 "Comparison of the SMAC, PISO, and Iterative Time-Advancing Schemes for Unsteady Flows", Journal of Computers and Fluids, Vol. 21, No.3, pp. 435-454.

Kim, S.-W. and Benson, T.J., 1993 "Fluid Flow of a Row of Jets in Crossflow - A Numerical Study", AIAA Journal, Vol. 31, No.5, pp. 806-811.

Koening, K. and Hodge, B.K., 1993, "A Suite of Personal Computer Programs to Support Propulsion Education", AIAA Paper 93-2053.

Mattingly, J.D., Heiser, W.H. and Daley, D.H., 1987, *Aircraft Engine Design*, AIAA Education Series, Washington, D.C.

Rand, R.H., 1987, "Computer Algebra in Applied Mathematics: An Introduction to MACSYMA", Pitman Publishing Inc.

Schlichting, H., 1968, *Boundary Layer Theory*, McGraw-Hill, New York, 6th ed.

Shapiro, A. H., 1961, *Shape and Flow*, Doubleday, New York.

Walatka, P.P, Clucas, J., McCabe, R.K., Plessel, T. and Potter, R., 1994, "FAST User's Guide", NASA Ames Research Center, NAS Division, RNP Branch.

Figure 1   Particle traces from vortex street video.
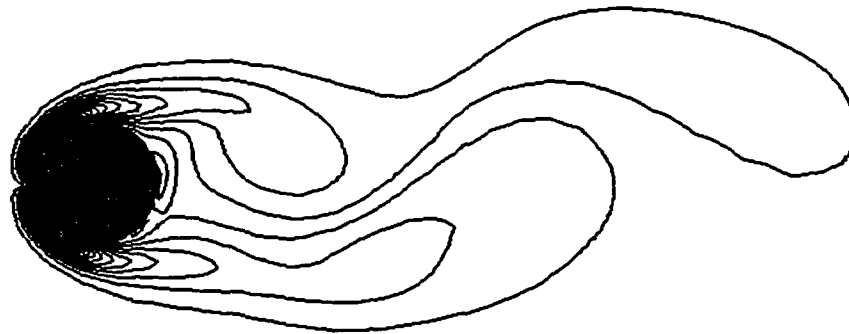
Figure 2   Vorticity contours from vortex street video.
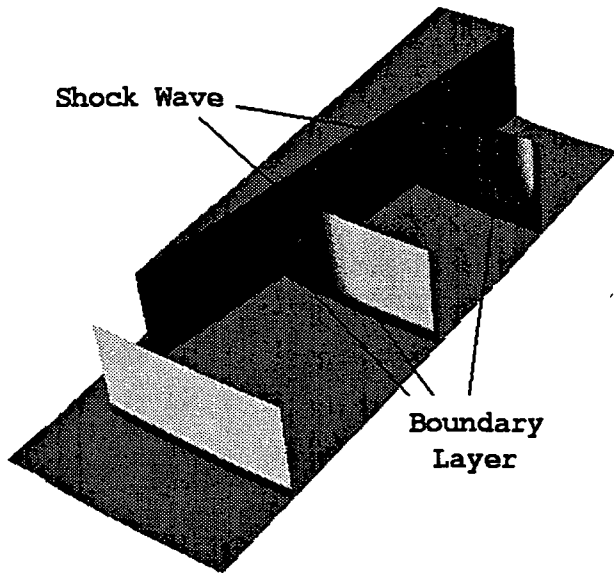
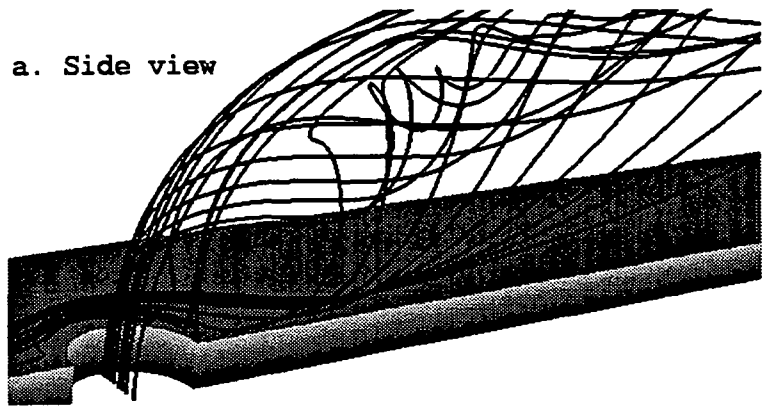Figure 3   Injected fluid visualization from video.

Figure 4 Mach number contours.

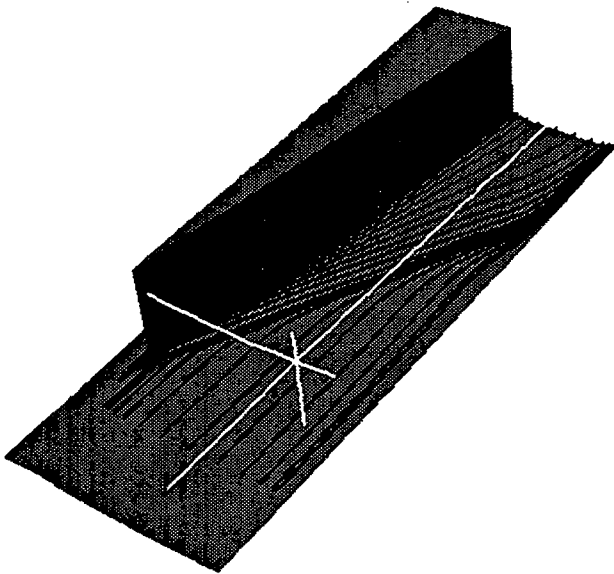a. Side view
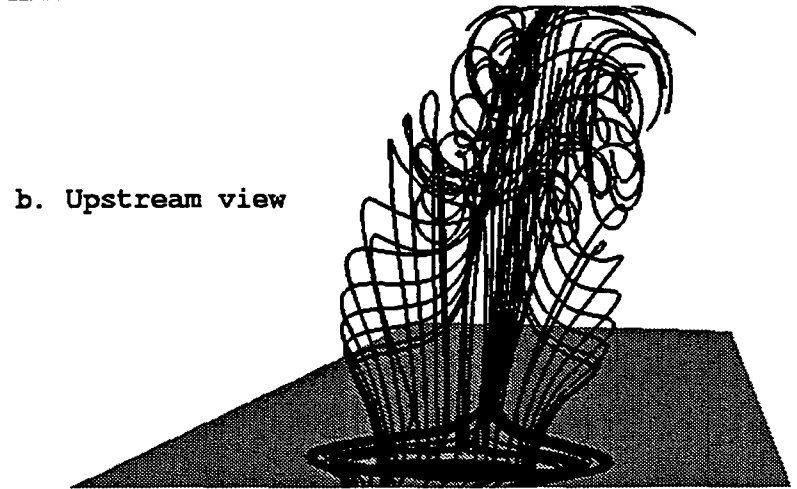
b. Upstream view

Figure 6 Particle traces of jet in cross-flow.
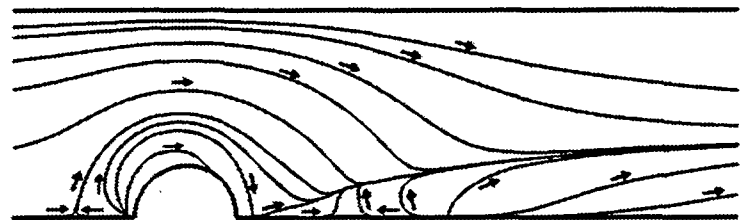

Figure 5 Surface oil traces.
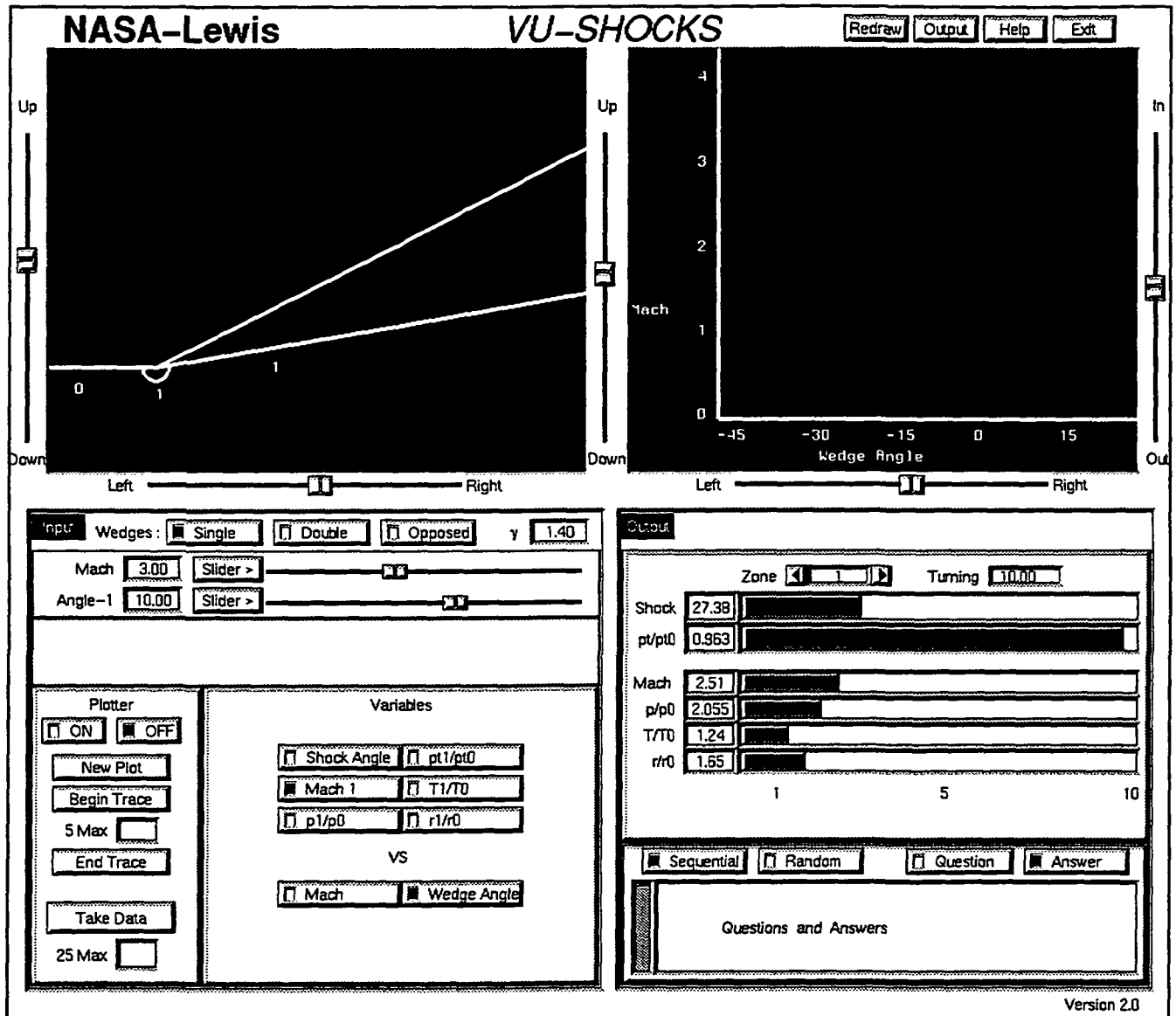

Figure 7 Surface topology of jet interaction.
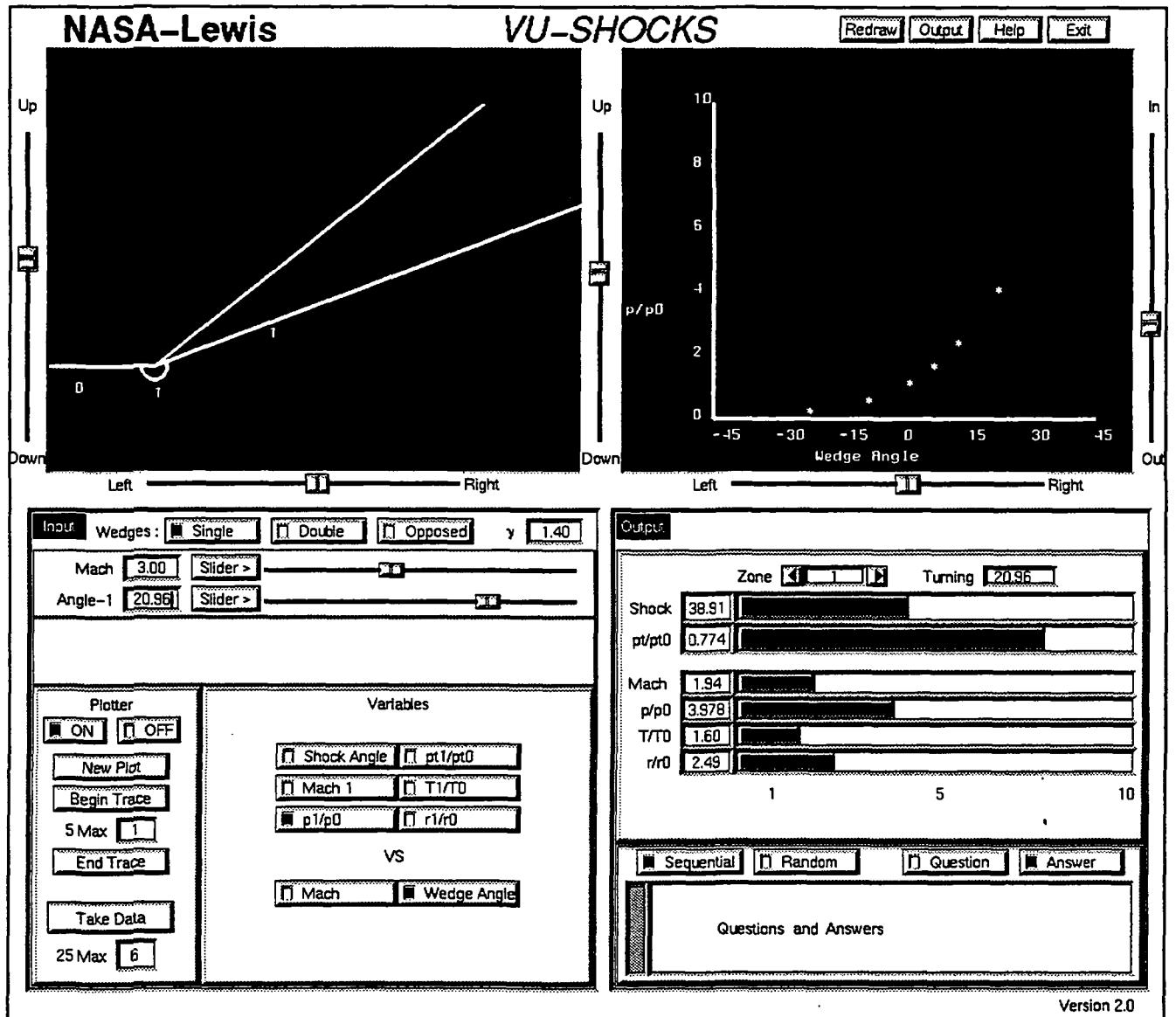
Figure 8   Screen dump of compressible flow simulator.
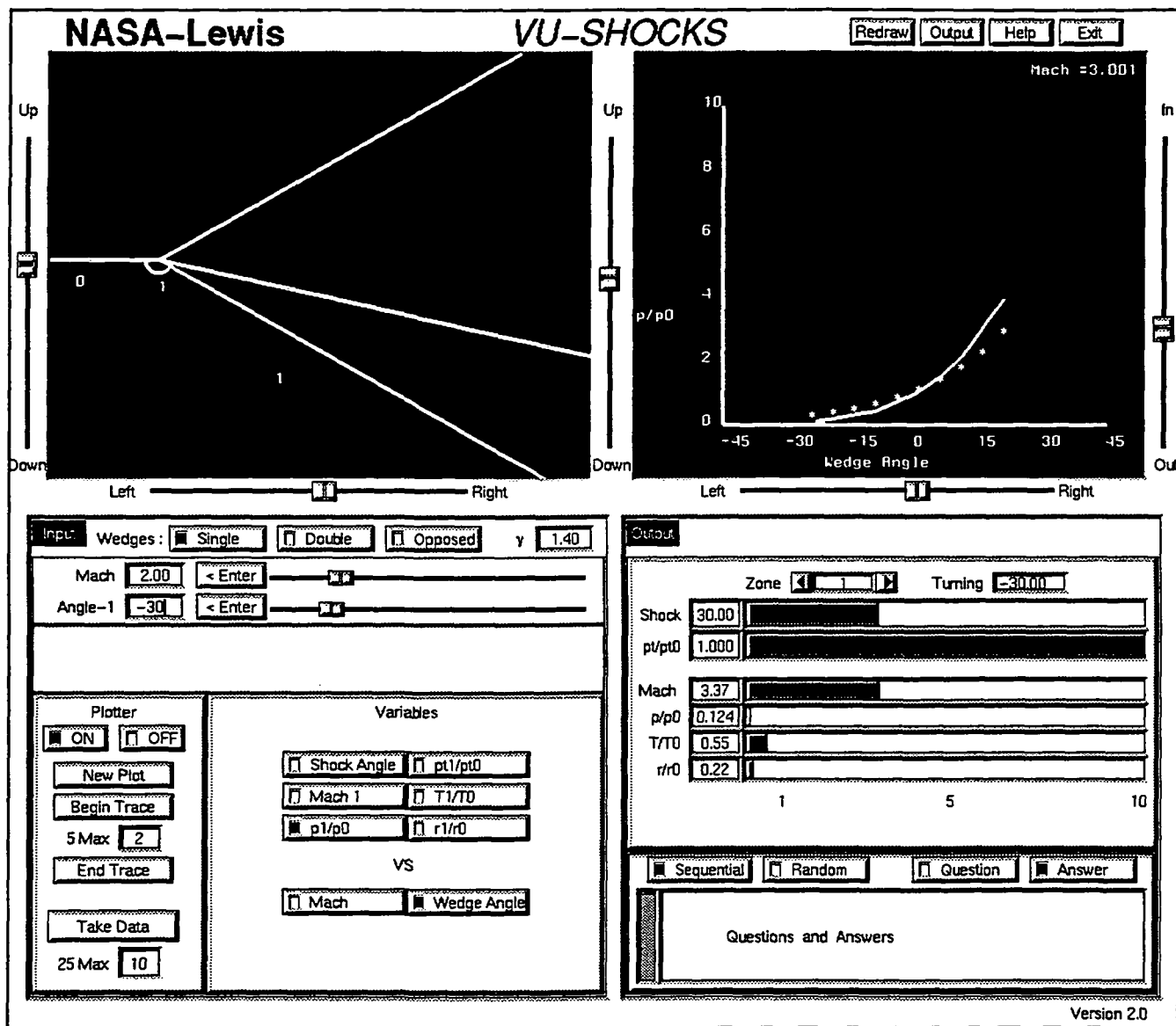
Figure 9 Acquiring data using the plotter.
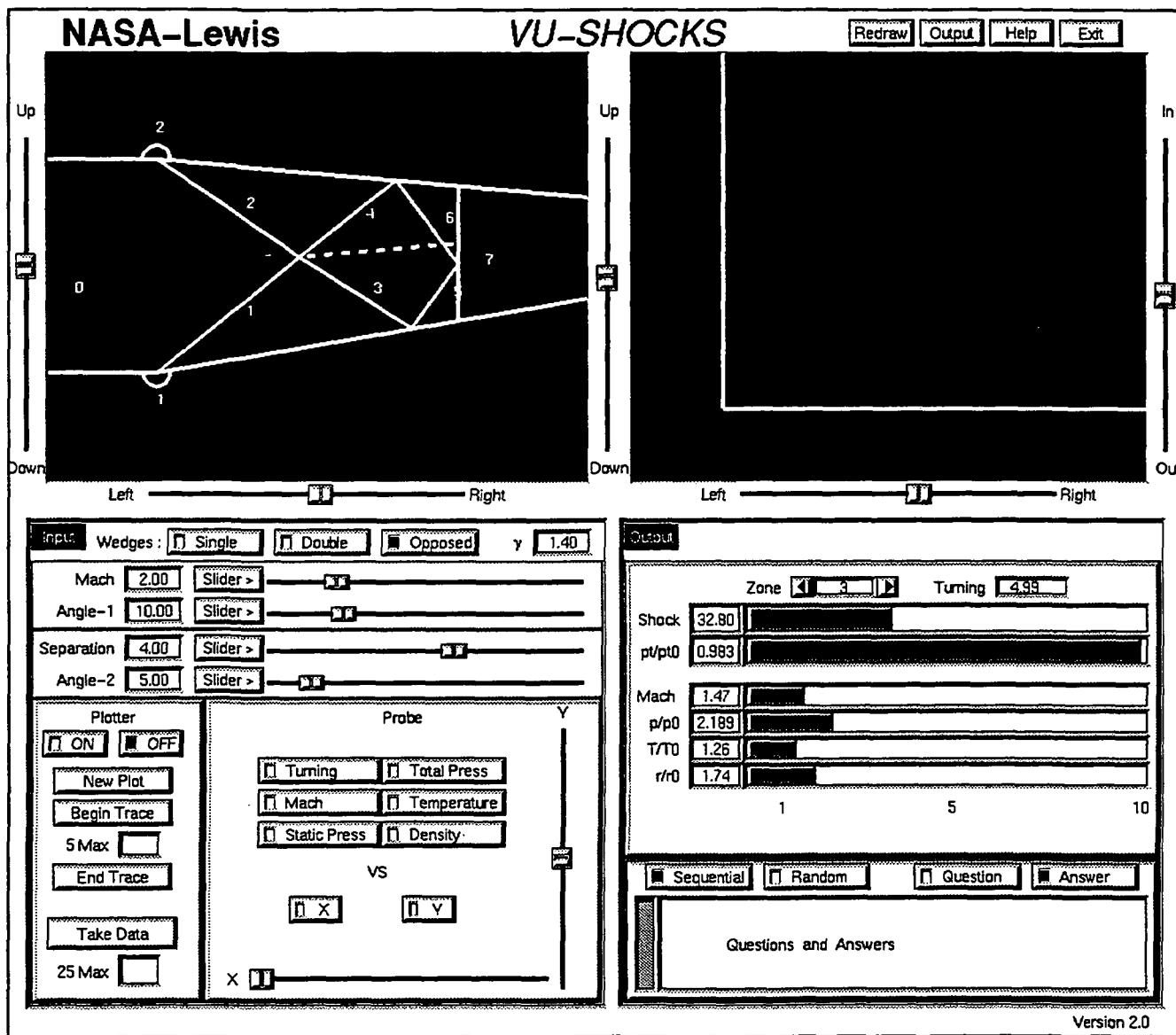
Figure 10   Completed plot.
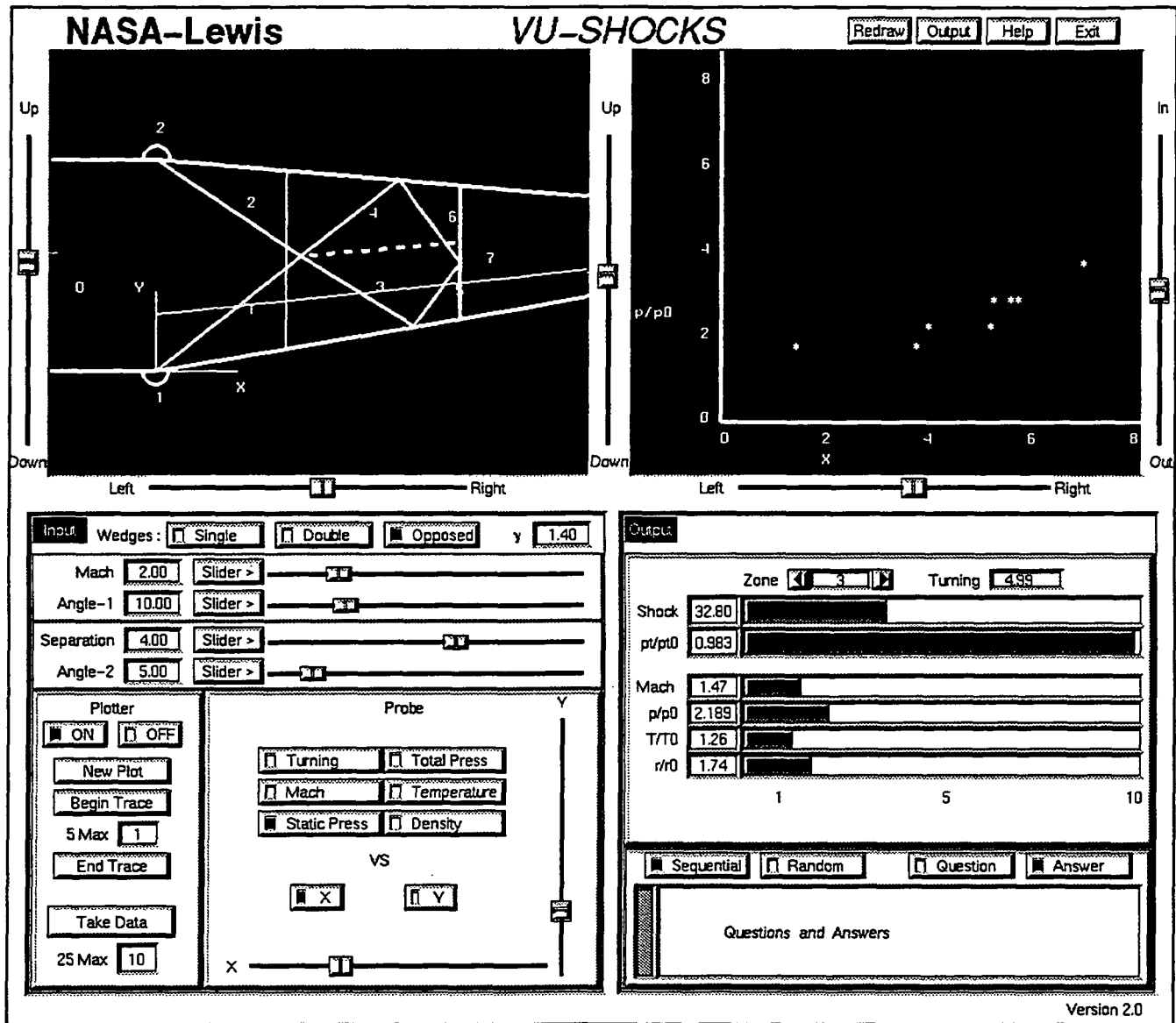
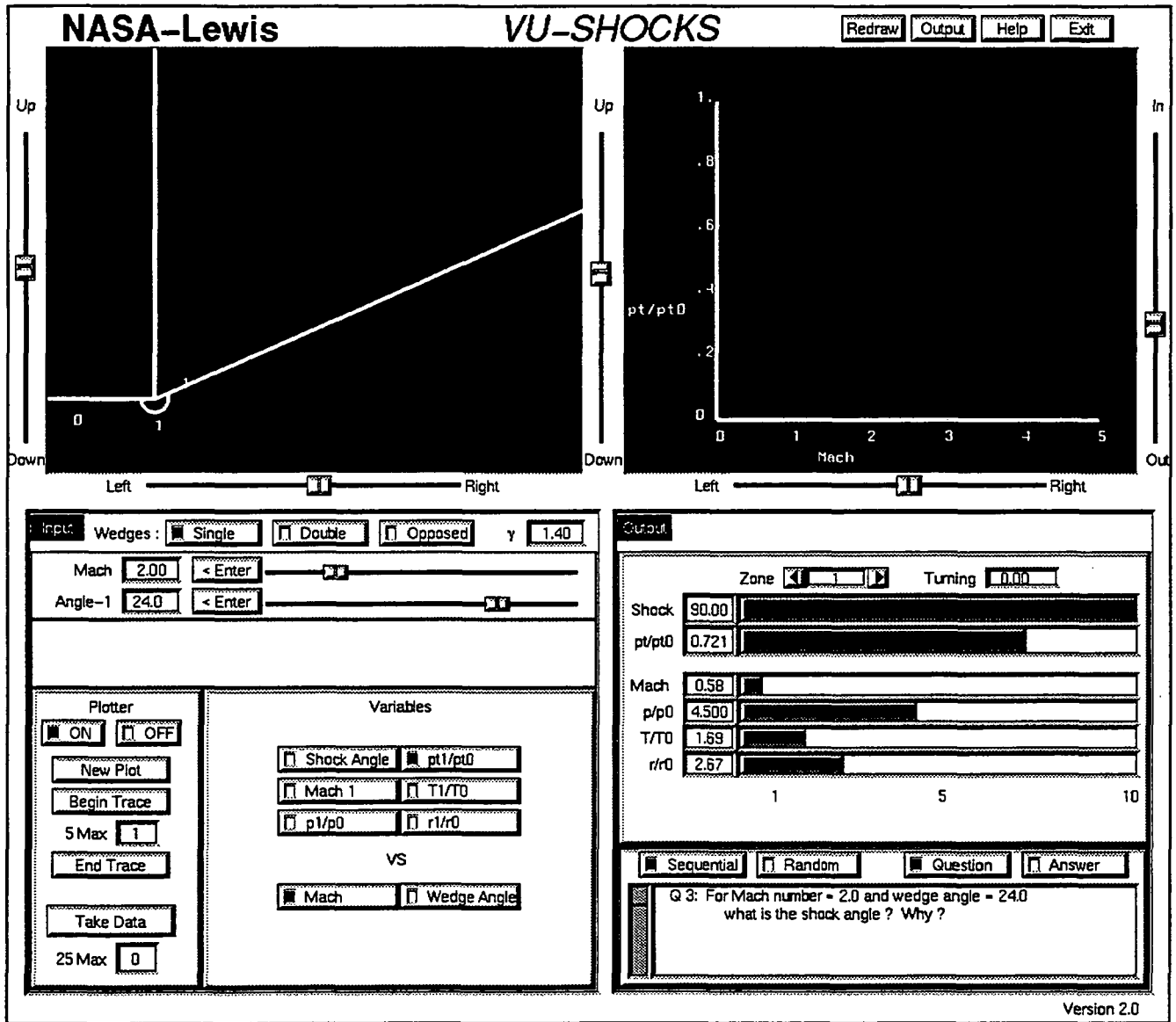**Figure 11   Crossed shock interaction.**

Figure 12   Data acquisition with "probe" .

Figure 13 Question and answer box.
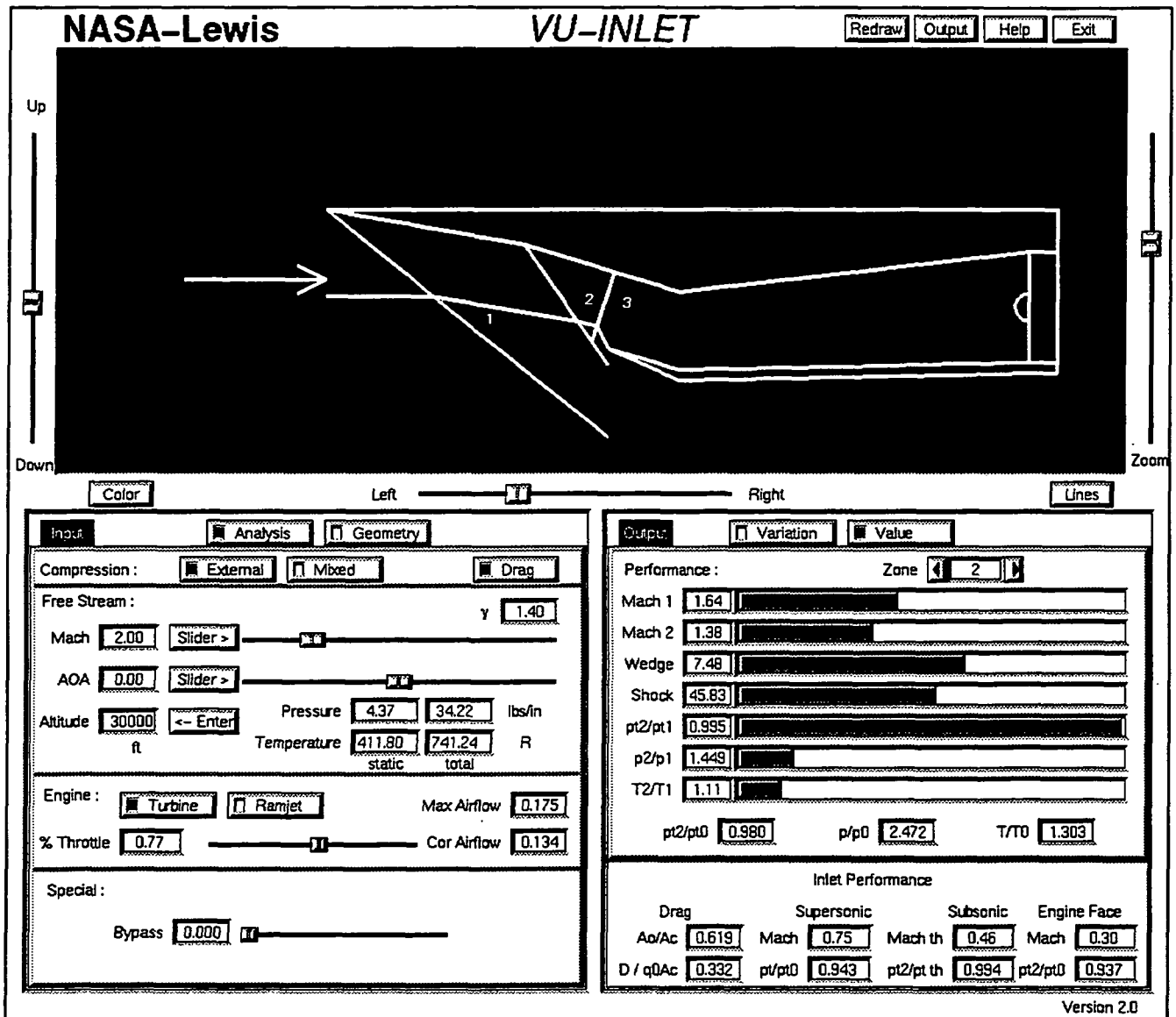
Figure 14   Supersonic inlet simulator.

REPORT DOCUMENTATION PAGE

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	June 1994	Technical Memorandum

4. TITLE AND SUBTITLE

On the Use of Computers for Teaching Fluid Mechanics

5. FUNDING NUMBERS

WU–505–62–52

6. AUTHOR(S)

Thomas J. Benson

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER

E–8851

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, D.C. 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM–106587

11. SUPPLEMENTARY NOTES

Prepared for the ASME Fluids Engineering Division Summer Meeting sponsored by the American Society for Mechanical Engineers, Lake Tahoe, Nevada, June 19–23, 1994. Responsible person, Thomas J. Benson, organization code 2670, (216) 433–5920.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 61

12b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)

Several approaches for improving the teaching of basic fluid mechanics using computers are presented. There are two objectives to these approaches; to increase the involvement of the student in the learning process, and to present information to the student in a variety of forms. Items discussed include: the preparation of educational videos using the results of computational fluid dynamics (CFD) calculations, the analysis of CFD flow solutions using workstation based post-processing graphics packages, and the development of workstation or personal computer based simulators which behave like desk top wind tunnels. Examples of these approaches are presented along with observations from working with undergraduate co-ops. Possible problems in the implementation of these approaches as well as solutions to these problems are also discussed.

14. SUBJECT TERMS

Educational; Software

15. NUMBER OF PAGES

18

16. PRICE CODE

A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102