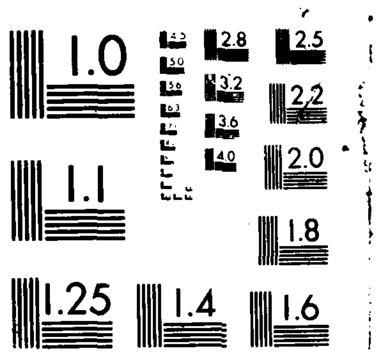
AD-A186 001 COMPUTATIONAL FLUID DYNAMICS REQUIREMENTS AT THE NAVAL 1/1
POSTGRADUATE SCHOOL(U) VRIJE UNIV BRUSSELS (BELGIUM)
C HIRSCH 01 OCT 86 NPS-67-86-007CR N62271-86-M-0243
UNCLASSIFIED F/G 20/4 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963 A

AD-A186 081

DTIC FILE COPY

(2)

NPS67-86-007CR

NAVAL POSTGRADUATE SCHOOL

Monterey, California

DTIC
ELECTE
OCT 22 1987
S D
D

CONTRACTOR REPORT

COMPUTATIONAL FLUID DYNAMICS REQUIREMENTS AT THE NAVAL POSTGRADUATE SCHOOL

CHARLES HIRSCH
VRIJE UNIVERSITEIT BRUSSEL
PLEINLANN 2,
1050 BRUSSEL, BELGIUM

OCTOBER 1986

Approved for public release; distribution unlimited.

Prepared for: Naval Postgraduate School
Monterey, California 93943-5000

87 10 1 368

CURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

. REPORT SECURITY CLASSIFICATION UNCLASSIFIED	1b. RESTRICTIVE MARKINGS
. SECURITY CLASSIFICATION AUTHORITY	3 DISTRIBUTION / AVAILABILITY OF REPORT Approved for Public Release; Distribution Unlimited
. DECLASSIFICATION / DOWNGRADING SCHEDULE	
PERFORMING ORGANIZATION REPORT NUMBER(S) NPS67-86-007	5. MONITORING ORGANIZATION REPORT NUMBER(S) NPS67-86-007

. NAME OF PERFORMING ORGANIZATION Prof. Dr. Ir. Charles Hirsch VRIJE Universiteit, Brussel	6b. OFFICE SYMBOL *(If applicable)*	7a. NAME OF MONITORING ORGANIZATION Naval Postgraduate School
. ADDRESS *(City, State, and ZIP Code)* Pleinlaan 2 1050 Brussels, Belgium		7b. ADDRESS *(City, State, and ZIP Code)* Monterey, CA 93943-5000
. NAME OF FUNDING / SPONSORING ORGANIZATION Naval Air Systems Command	8b. OFFICE SYMBOL *(If applicable)* AIR 931E	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER N62271-86-M-0243/N0001986WR61270

. ADDRESS *(City, State, and ZIP Code)*

Washington, D.C. 20361

10. SOURCE OF FUNDING NUMBERS

PROGRAM ELEMENT NO.	PROJECT NO.	TASK NO	WORK UNIT ACCESSION NO.
61153N	WR024	03	001

1. TITLE *(Include Security Classification)*

Computational Fluid Dynamics Requirements at the Naval Postgraduate School.

2. PERSONAL AUTHOR(S)
Hirsch, Charles

3a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT *(Year, Month, Day)*	15. PAGE COUNT
Final	FROM 7/86 TO 9/86	1986-October 1	55

6. SUPPLEMENTARY NOTATION

7. COSATI CODES

FIELD	GROUP	SUB-GROUP

18. SUBJECT TERMS *(Continue on reverse if necessary and identify by block number)*
Computational Fluid Dynamics (CFD)
Naval Postgraduate School

9. ABSTRACT *(Continue on reverse if necessary and identify by block number)*

The needs for CFD in connection with student and faculty activities in the engineering departments at NPS are reviewed. Emphasis is placed on internal, propulsion related flows. Currently available CFD codes are also reviewed and computer hardware and services are evaluated. Conclusions and recommendations are made for future directions.

20 DISTRIBUTION / AVAILABILITY OF ABSTRACT ☒ UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT ☐ DTIC USERS		21 ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED
2a. NAME OF RESPONSIBLE INDIVIDUAL R. P. SHREEVE	22b. TELEPHONE *(Include Area Code)* (408)646-2593	22c. OFFICE SYMBOL 67Sf

D FORM 1473, 84 MAR

83 APR edition may be used until exhausted.
All other editions are obsolete

SECURITY CLASSIFICATION OF THIS PAGE

U.S. Government Printing Office: 1986—606-24.

TABLE OF CONTENTS

A-1

A. Review of engineering needs for CFD, related to propulsion oriented internal flows, in connection to student and faculty research activities at NPS.

Part I. Engineering needs for CFD in relation to student activities

The propulsion related courses are briefly reviewed with regard to their content in connection to the impact of fluid dynamical phenomena on the properties, behavior, operation and design of propulsion systems.

Most, if not all, of the propulsion systems for aircraft , missiles as well as ships are based on complex flow mechanisms and energy exchanges. The understanding of these complex phenomena is essential to the comprehension of the operational characteristics of a propulsion system; to the awareness of the mechanisms dictating its performance as well as limiting its efficiency and the potential for improvement.

A thorough understanding of the flow related properties which condition essential design parameters, allows us to assess the potential prospects of new design options and possible areas for improvements.

The advent of high speed computer facilities, coupled with highly efficient algorithms, has rendered the application of Computational Fluid Dynamic simulations of complex flow systems accessible to the integration of advanced design and analysis methodologies.

This capability can and should also be used at the **educational** level, in the form of visualizations of numerical simulations of basic, flow phenomena . This will also enable the student to investigate, through the graphical analysis of pre-stored computational results on adapted work-stations, the influence of geometrical, operational and environmental parameters on the properties and behavior of propulsion systems and their components.

The present investigation aims at defining the ways these goals could be achieved, in regard of the specific needs, of the available codes, of the required hardware, and incorporation in the present scheme of laboratory or demonstration activities for the students.

The following, brief review of the fluid dynamic content of the propulsion courses, includes also related courses dealing with external aerodynamic problems, in so far as they deal with essential aspects related to the propulsion systems, such as, for instance, engine airframe integration of aircraft, or missile aerodynamics as it affects control strategies and performance.

The review of the courses has been restricted to the graduate program, excluding the preparatory courses from the present investigation of CFD applications.

Presently very few of the above mentioned courses do apply computer based analysis or design al the level of the student laboratory work. As an example, students do generate their own codes on personel computers for compressor design work or make use of an existing interactive program for turbine design investigations.

These examples of programs are based on simple one dimensional models. It is thought that much benefit would be gained in the comprehension of basic effects and mechanisms, if the students would have access to the results from higher level codes, which simulate in a more accurate and complete way the internal or external behavior of the flow.

Let us illustrate this by some practical examples:

Design and analysis of compressors and turbines.

A program is already available on the NPS mainframe, allowing a Quasi-three dimensional analysis of multistage compressors and turbines (program Q3DFLO-81).

The input preparation of this code might not be appropriate within the students laboratory activity. However typical, simplified designs of the type performed by the students can be analysed separately and prepared for visual investigation by the students.

For instance, the complete hub-shroud and blade to blade distributions could be displayed on a graphic station for different mass flows, allowing the student to develop an improved understanding of the detailed flow behavior inside the blading for varying incidences, to familiarize with the effects on loss generations, end wall flow properties, blade loadings of different design options or criteria.

Full three-dimensional analysis of a blade row

By having access to full three dimensional, viscous,transonic flow simulation codes for compressor or turbine blade rows, the results of such a large scale computation can be stored and displayed on a graphical station for detailed visualizations.

With the graphical engineering workstations presently available on the market, and soon to be accessible in the CAD/CAE laboratory within the Departments of Aeronautics and Mechanical Engineering, representations of the complex flow pattern

obtained from the computation can be analysed through animated computer simulation.

This allows the student to familiarize himself with fundamental flow structures as they affect the internal energy exchange and the generation of three dimensional losses, such as those induced by horsehoe vortex structures in high turning turbine nozzles, by three dimensional shock structures in inlet fans of high performance engines, by tip leakage flow patterns, by part span dampers, etc...

In a second stage, students might be called upon to apply interactively graphical simulation methods in order to extract particular flow properties from the stored data, simulating hereby the results of experiments, displaying pressures, temperature fields, flow angles and velocities in various planes or cross-sections, loss distributions and other variables of interest to the understanding of design and operational criteria.

Unsteady flow patterns in engines, inlets, exhausts, combustors

Considerable educational benefits can be obtained from the analysis of unsteady flow phenomena as they affect the operation of engines, inlets, missile control, etc.

Many operational problems are caused by spontaneously generated flow instabilities, such as the so-called inlet buzz instability, observed in supersonic inlets under certain conditions. Spectacular computer simulations of these phenomena have already been performed and visualized.

Similarly, the internal unsteady flow in combustors can be visualized allowing the investigation of effects such as thermal distributions, flame propagation and instability, influence of turbulence, and so on.

SUMMARY

By making use of the availability of large scale computational fluid simulation codes, applied to internal and external flow aspects dealt with in the various courses related to propulsion, a large data-base of pre-computed results can be established for subsequent use by the students.

Through display on graphical stations, the students will have the opportunity to gain a deeper understanding of the basic and often complex flow mechanisms affecting the operation and design criteria of aircraft, rockets and missiles.

In a second stage, students could generate their own visualized flow representation on

The introduction of the above mentioned applications, in the form of active demonstrations of fundamental operational properties of propulsion systems, requires graphical stations. Depending on the degree of complexity of the investigated flow system, high end products, such as high level graphical stations to be available within the CAD/CAE laboratory, might have to be used. For less sophisticated problems, personel computers might be considered, provided they are interconnected in a network with the main units containing the basic data.

Part II. Engineering needs for CFD in relation to research activities

General considerations

Numerous research activities in the field of propulsion, developed at NPS and connected to internal and external flow properties, rely to various degrees on Computational Fluid Dynamics.

Most of these activities however call upon flow simulation codes as support of broader investigations, either of experimental or theoretical nature. Fluid flow analysis is applied to complement experimental programs, by providing coherence and, or, accuracy cross checking. It is also used to provide information on quantities not directly measured or to gain some additional knowledge by extending numerically the range of certain experimental variables.

Flow simulations are applied as well in the preparation, design and development of new experiments, new facilities and design/development of advanced components of propulsion systems, reducing hereby the amount of experimental preparatory testing.

It is therefore safe to consider that in a very large measure, the activities connected to CFD at NPS are essentially oriented towards the use and application of existing flow simulation codes, as opposed to the activity of code development.

A large effort is nevertheless performed by some groups in the adaptation of existing codes to their specific needs, by extending the original range of application or by modifying some parts of the program to match other geometrical and environmental conditions.

Part II. Engineering needs for CFD in relation to research activities

II.1. Turbopropulsion research

Cascade test rig

The experimental research work on the subsonic cascade test rig is substantiated by extensive blade to blade computations, presently at an inviscid, two-dimensional level.

These computations have played an important role in assessing the accuracy and consistency of the data.

The addition of viscous components into the computation, either through a viscid-inviscid interaction approximation or through a full Navier-Stokes two-dimensional code, would allow the investigator to take into account the influence of viscosity on the pressure distribution, to predict separation points and to estimate losses and their variation with incidence. This prediction capability would provide a support for assessing the accuracy and reliability of the measurement techniques and procedures.

In addition, it is in the line of one of the purposes of the experimental program, namely to provide data for validation of flow prediction codes.

Transonic Compressor test rig

The transonic test rig allows very fine investigations of complex flow phenomena typical of high performance engine components. Investigations of unsteady aerodynamic effects, of shock configurations and shock-boundary layer interactions, three dimensional flow fields and their effect on the loss generation mechanisms, can be performed via a unique combination of sophisticated measuring probes and data reduction techniques.

The data have been compared up to now with computations provided by a quasi-three dimensional computer program, based on an axisymmetric approximation, providing a simplified view of the three dimensional flow field and estimating the losses via empirical correlations.

Much would be gained by the availability of a full three dimensional, viscous program allowing to predict phenomena such as spontaneously generated unsteady effects due to shock-boundary layer interactions or three dimensional stagnation pressure distributions and shock structures. This would here again provide directions, guidelines to the separation of the various sources and effects determining the measured distribution of flow variables. It would also allow the investigator to increase and improve further the confidence in the calibration and data reduction methodology of the complex set of probes.

Wave rotor research

A unique test rig for investigating new directions for energy exchange in gas turbine engine components, through unsteady wave processes is operated at NPS.

Due to the advanced and innovative character of the associated research project, large portions of the project development rely on the computational prediction capability of supersonic flow configurations with strongly interacting shockwaves and contact discontinuities. This requires flow prediction codes with very high resolution in the prediction of unsteady discontinuities and shock reflections in two dimensional, and eventually three dimensional, rotating geometries.

II.2. Solid Fuel Ramjet Combustion

Studies of solid fuel ramjet combustion processes are conducted in a long range effort to develop these systems for air and surface launched missiles. In parallel with the experimental investigations, computer flow modeling including chemistry is an essential part of the overall effort to determine the effect of design and operational parameters on the achievable performance.

The use and availability of flow simulations of the combustion process is an indispensable tool for the prediction of the effects of the air injection systems and combustor geometry on the fuel utilization and on the internal flow field. This requires three-dimensional flow computations in complex geometries, taking into account the effects of turbulence, chemistry kinetics and radiation.

II.3 Other research topics at NPS with strong needs for CFD applications

We list here other representative research topics currently pursued in the Departments of Aeronautics and Mechanical Engineering, having a strong involvement in the use of CFD codes.

- Exhaust of supersonic/subsonic jets
- Fire safety investigations in aircraft and submarine enclosures (study of smoke propagation)
- Unsteady flow through cascades with vibrating blades in flutter applications
- Heat transfer in stagnation region of turbine blades
- Natural convection in enclosures for electronic cooling applications.

These topics are illustrative of research programs which rely almost entirely on the availability of high level, three dimensional flow prediction codes.

The CFD needs arising from the above mentioned research projects can be fullfiled in two different ways, taking into account that the research activities are directed towards the development and investigation of operational systems or components, and not in the development of basic CFD codes.

i) Application of existing commercial codes

This approach has the advantage of putting at the disposal of the user an extensive documentation under the form of User's Guides and instruction manuals. In addition, the commercial organisation will provide regular updatings of the programs.

On the other hand the programs are generally delivered without the source, as a "black box", preventing the user from adapting the codes to his own specific needs, or extending them to cover his field of application more completely.

ii) Application of research codes developed at other institutions.

This approach offers the availability of open codes, with communication of the source, allowing adaptation, extension, modification to fit the specific purposes of the research.

On the other hand these programs are generally very poorly documented and difficult to use without the assistance of the original developer of the code.

In this connection, the recently created Joint Institute between NPS and NASA AMES Research Centre will provide a framework through which part of the above mentioned problems could be solved, since NASA AMES is a major developer of general CFD codes.

Summary

The needs for CFD codes arising from research activities can be covered to a large extent by existing codes of either commercial type or by research codes developed at other institutions, for instance at NASA AMES Research Center.

Commercial codes are generally better documented but cannot be modified nor accessed for adaptations or extensions to meet specific needs arising from a research project.

Research codes are generally poorly documented and require mostly the support of the original developers. In this connection, the NAVY-NASA JOINT INSTITUTE OF AERONAUTICS offers a unique framework of communication, since many CFD simulation codes developed at NASA AMES could be used at NPS in support of research activities (refer to part B for specific descriptions). The technical support needed for adaptation and extension of existing codes to meet specific requirements of the investigation could be provided within the Joint Institute's activities.

Conclusions to Part A

1. There is a need for the availability of a range of fluid simulation codes ranging from codes with a general application domain for full three dimensional analysis, to codes based on simplified models and for more specific applications .

2. There is a clear need, expressed at the level of Faculty members of the Aeronautics and Mechanical Enginering Departments for organized support for the use, access and operation of CFD codes, with regard to input and output file management and analysis on high performance engineering and graphical workstations.

3. A general concern has been expressed by most of the users of CFD codes for the easy access and availability of interactive graphic stations for flow data analysis.

4. Codes of the first type, for general purpose applications, should be supported by dedicated personnel of one or two persons, in order to ensure the continuity of the initial efforts of installation, to guarantee the maintenance and upgrading to new versions or to additional improvements provided by the originators of the codes, to maintain the appropriate documentation, and to provide support to new users.

   This support team should operate and maintain the access to the codes, which could be run either on the mainframe of NPS or on external supercomputers, via the appropriate network connections.

   The support team should also ensure the connection with the adapted hardware, graphical stations for instance, in order to provide the straightforward transfer of output files and appropriate network connections.

   The support team should maintain the availability of stored output data files prepared and developed for the support of student courses.

   This last aspect is essential in order to allow the access of the students to the analysis and visualization of flow systems in relation to associated courses .

5. The installation of a CFD support team could be considered in relation with, or as part of, the CAD/CAE laboratory installed jointly by the Departments of Aeronautics and Mechanical Engineering. This seems appropriate since Faculty members of both Departments have expressed a strong interest for the above

mentioned organization and management of available, or to be installed, general purpose codes.

6. The more specific codes would best be managed by the laboratories or Faculty members concerned, with regard to their research needs.

However, the accessibility to efficient and operational output analysis workstations is essential in the efficient use and application of high level codes. The smooth transfer of input and output files from a local terminal could be facilitated by the support of dedicated personnel.

The access of the students to prepared output files for demonstrations, analysis and visualizations could be organized at the CAD/CAE laboratory with the active participation of the support team, or locally within the concerned laboratory when the appropriate equipment is available.

B. Review of available codes at NPS and other Governmental Organisations for viscid and inviscid flow predictions.

Part I. Codes presently available at NPS for aeronautical applications

The codes listed in Appendix A cover internal and external aeronautical oriented flow configurations.

Specific applications connected to atmospheric flows, ocean circulation models, weather prediction codes are not reported here, since they are generally not adaptable for flow configurations of aeronautical interest. However the potential applicability of certain codes, to be listed in the following, to these fields will be mentioned when appropriate.

The presently available codes at NPS can be subdivided into the following subgroups:

i) General purpose codes

Under this group one can consider the codes

PHOENICS
Q3DFLO-81
GARRETT/AiRESEARCH COMBUSTION Code
UNDSAFE

ii) Specialized internal (turbomachinery) flow codes

Under this group one can list the codes

BLAYER
TSONIC/QSONIC
MERIDL
EULER 1
MESHGEN/TURBO
SUPERSONIC FLOW IN OSCILLATING CASCADES
UNSTEADY, INVISCID FLOW IN CASCADES

iii) Specialized external flow codes

Under this group ons can list the codes

KELLER BOX METHOD FOR BOUNDARY LAYERS
VISCID-INVISCID INTERACTION ON AIRFOIL
FLOW OVER WING-BODY JUNCTION
PANEL METHOD

COMMENTS ON LISTED GENERAL PURPOSE CODES

PHOENICS: The PHOENICS code is generally of application in the subsonic range and is of particularly interest in presence of heat transfer and turbulence effects. It contains a two equation model for turbulence and applies to three dimensional flow configurations. Although the originators of the code confirm its applicability to supersonic flow regimes, it is to be noted that the shock capturing properties are rather poor .

Its generality requires an initial time investment from the potential user, in order to familiarize himself with the use of the code, the input geometry being introduced via a set of subroutines to be written by the user.

The existence of a support team, able to provide assistance to a new user in accessing the code and in the file management at the input level as well as at the output level, would greatly enhance the effectiveness of the initial efforts of installation and operation of the code.

Q3DFLO-81: This code is of application for general single or multistage turbomachines, compressors or turbines. It contains separate modules enabling to compute axisymmetric duct flows, multistage through flows or quasi three-dimensional blade-to-blade flows. The through flow computations allow for supersonic relative velocities, but the blade-to-blade flows are limited to the subsonic range.

End wall boundary layers can be computed by an integral method in interaction with the main through-flow.

GARRETT/AiRESEARCH COMBUSTION Code: This is a general three-dimensional viscous code for steady, subsonic flows, containing a two equation model for turbulence but its main feature and domain of application lies in the domain of combustion, in particular gas turbine combustors. It includes finite rate chemical kinetics, a six-flux model for radiation, soot emission and liquid spay options.

UNDSAFE: This code is developed for turbulent buoyant flows in enclosures with vents and designed for calculating the spread of fire and smoke. It applies to two dimensional, viscous, subsonic flows and contains an algebraic model for turbulence. Its essential features are the inclusion of strong buoyancy, thermal stratification effects and fire plus smoke source simulations.

COMMENTS ON SPECIALIZED CODES

The turbomachinery and internal flow codes cover essentially inviscid flow simulations, eventually corrected by external loss sources or by boundary layer calculations. In addition, the spatial approximations are limited to a quasi three dimensional representation, connecting axisymmetric streamsurfaces with an averaged through-flow in a meridional section.

There is clearly a need for complementing the available flow prediction capability by a full three-dimensional, viscous flow code enabling a prediction of the complete flow properties in a rotating blade row, including the effects of turbulence and rotation.

The available external flow codes are limited to potential flow models, which are known to be of limited validity in transonic flows with shocks. A wellcome extension of the prediction capability of external flows along aircraft components and missile bodies, would be achieved by inviscid Euler codes and eventually by Navier-Stokes, thin shear layer model programs.

The available boundary layer codes cover the two-dimensional flow domains and an extension of interest, which could be coupled to the inviscid models, would be the availability of a three-dimensional boundary layer program.

Part II. Codes for aeronautical applications available at other Institutions

A selected number of codes available from other institutions and which might fulfill some of the CFD needs of research and education at NPS are listed and referred to in the following.

The principal consulted sources are the COSMIC catalog and NASA Research Centers, more particularly NASA AMES Research Center. The reason behind this latter restriction is twofold:

i) most of the important and up to date codes are available at NASA AMES

ii) the creation of the JOINT INSTITUTE between NPS and NASA AMES provides a unique framework for the transfer of codes, including the major aspects of documentation, support in implementation and follow-up by the originators of the codes.

The contact person at NASA AMES for CFD activities in relation with the Joint Institute is Dr. T. Holst, Chief of Applied Computational Fluids Branch.

It is to be noticed that the NASA Research Centers do not dispose of a listing of their available codes. The following information is an outcome of direct contacts with different groups at the various Centers and from a survey of the published literature.

II.1: Codes available through COSMIC

The COSMIC organisation has issued a collection of computer program abstracts for aerodynamics applications, the content of which is listed under Appendix B

Due to the time delays from the generation of a code to its appearance on the COSMIC catalog, some of the available codes might not be representative of the latest state of the art. It is therefore suggested to check the date of program generation as well as the methodology applied.

The impact of a non up-to-date code might represent an order of magnitude in computer time and cost in comparison to more efficient codes.

II.2: Codes available through NASA AMES

A few general codes, representative of the latest developments in computational fluid dynamics have been selected and are summarized in Appendix C.

ARC2D/ARC3D: These codes have been extensively tested and optimized and can be considered as extremely reliable. They solve the thin shear layer approximation of the Navier-Stokes equations for two and three-dimensional airfoils and wings and contain a simplified turbulence model.

The adaptation and extension of these codes to three dimensional internal flow configurations and turbomachinery blade rows should be considered as a worthwile activity and investment for future needs, both in education and research.

TAIR: This code is based on the full potential equation and is suited for flows along airfoils, wings and wing-body combinations in flow situations where the potential approximation might be considered as valid, that is roughly for Mach numbers below 1.25.

Other potential codes for more complex configurations are available from NASA LANGLEY Research Center.

TURF: This code extends somewhat the domain of application of ARC2D, by providing more options with regard to the turbulence models, the possibility of treating axisymmetric problems as well as allowing also calculations with the full Navier-Stokes equations next to the thin shear layer approximation.

In addition it offers an internal mesh generation possibility and incorporated plotting options.

INS3D: This code is developed for stationary, incompressible, three dimensional flows and can be applied to internal as well as external flow systems. It contains a two equation turbulence model.

It has been tested and applied industrially to complex configurations such as the hot gas manifold of the Space Shuttle main engine as well as many other flow configurations. It could be recommended for complex incompressible flows and would provide an adequate basis for flows around submarines for instance.

STATOR-ROTOR INTERACTION CODE: This code is of a more specialized nature, having been developed for a specific application. It provides however highly interesting results for the very difficult but important problem of stator-rotor interactions in compressors and turbines.

OTHER CODES AVAILABLE FROM NASA AMES: More specific programs, directed towards particular applications do exist at various divisions of the Center. Since no detailed information is available, the following information can be given for specialized applications.

Helicopter flows:	Applied Fluid Dynamics Division
Flow in inlets:	Aerodynamic Division
Non Equilibrium flows:	Thermal Sciences Division

II.3. Codes available from other Institutions

The following Institutions have developed high level codes which might be of interest to research and educational activities at NPS.

i) Wright Patterson Air Force Base- Air Force Wright Aeronautical Laboratories

Some of the most advanced applications of the full Navier-Stokes codes to complete three dimensional aircraft configurations have been developed at this Institution by Dr. J. S. SHANG and coworkers.

Among the most interesting applications, calculations of the spontaneously generated inlet buzz in supersonic inlets and of the unsteady flame in an axisymmetric combustor can be mentioned.

These two last computations have been summarized in a movie which is obtainable from Professor J. S. SCOTT at University of Dayton Research Institute, Dayton, Ohio.

ii) NASA LEWIS Research Center

The codes of the ARC2D/ARC3D family have been adapted by R. CHIMA to two dimensional cascades, and applied to various compressor and turbine bladings.

Some three dimensional extensions and developments are in progress.

iii) NASA LANGLEY Research Center

Codes similar to those available from NASA AMES can be obtained from NASA LANGLEY. They are generally based on different numerical algorithms but cover the same range of general applications.

However, for very specific applications, particularly for external flows, it might be advisable to inquire with the Transonic Aerodynamics Division, the Low Speed Aerodynamics Division or the High Speed Aerodynamics Division as principal users and developers of aeronautical oriented codes.

C. Evaluation of available computer hardware and services at NPS with regard to the preparation and analysis of large scale 3D flow simulations.

Part I. General requirements for large scale computations in support of Research and Education

Large scale flow simulations can be performed on a variety of computers, from mini- to supercomputers, depending on the scale of the problems and the available time.

Adaptation of codes, debugging, input data file preparation, mesh generation should generally be performed on local computers, mainframes or engineering work stations.

For large scale simulations required by research projects, the prepared input files will have to be sent to a large size computer, most probably a supercomputer. After execution of the computations, the output files have to be transferred to the user for analysis and display. This stage is generally recognized, by all the present user's of three-dimensional flow computations, as the most essential one in the effective use and exploitation of the flow simulations. It requires the availability of high level graphical stations, with high resolution and color display, allowing real-time dynamic visualisations, continuous rotation and zooming of the generated visual representations.

It should be noted that this capability is also essential for the generation and analysis of three-dimensional mesh geometries.

Therefore the requirements of large scale flow simulations call for an effective operational network, connecting on one end the NPS mainframe computer to external supercomputers and on the other end connecting local workstations to the general network for exchange of input and output files. The workstations should contain a certain number of high resolution, color graphic stations allowing near real time 3D representation and a rate of 3D vector clipping and transformation of the order or higher than 80000 per second.

The selected workstations should be able to run the remarkable 3D color plotting utility program developed at the Computational Fluid Dynamics Branch of NASA AMES by P. Buning. This code, called PLOT3D, is designed for the static and dynamic graphical representation of three-dimensional flow data and includes options for animated visualisations which are of great didactic value.

With regard to the educational aspects of the applications of CFD to various propulsion and aeronautical oriented courses, considerable benefit would be obtained by allowing the students to analyse on local PC's or workstations graphical outputs of previously performed computations.

This could be realized optimally by networking the existing PC's within the Department

messages between instructor and students. The local area network of the Joint AERO-ME CAD/CAE Laboratory should be the main basis for this extension.

Part II : Network Systems at NPS

The computer systems at NPS have presently the necessary potential for the development of an extended and operational network, both external and local.

External network.

Figure 1 represents the present basic external network system at the Computer Center of NPS. It is centered around the Interface Message Processor (IMP) which is connected to the Defense Data Network (DDN) as well as to the EARN/BITNET network. Present access is through telephone line connections, via terminals monitored by a Terminal Access Control (TAC) system and several external computers can be accessed presently through these terminals.

The VAX minicomputer of the Departement of Computer Sciences is connected to the IMP.

The extensions, foreseen by the end of this year at the Computer Center, are the connection of the IBM 3033 mainframe to the IMP, enabling the access of the DDN via the local IBM terminals.

Since the Joint Institute between NASA AMES and NPS offers a privileged framework for cooperation in research and education, priority should be given to a strong connection between the NPS computers and the NASA AMES supercomputers, enabling fast transfer of files in both ways. This requires the development of a File Transfer Protocol (FTP) compatible at both ends of the connection, with the necessary software support for full interactive exchange.

In order to achieve this goal contacts should be established between the Computer Centers of the two institutions.

Figure 2 shows the extended network configuration, with the NASA AMES computers connected via DDN.

The supercomputer of the Naval Environmental Prediction Research Facility (NEPRF), a CYBER 205, is also connected via the IMP to the NPS Computer Center. The conditions for accessing this computer must be discussed, in connection with the research project involved, with the responsible authorities of NEPRF.

Internal local network.

several graphical workstations. A recent development is the inclusion of a cluster of Personal Computers into the network, through a single concentrator. The technical details of the PC network incorporation into ETHERNET are described in the Master Thesis of R. L. Hartmar and A. F. Yasinsac, Dept. Computer Sciences, June 1986.

The development of a similar system in the Departments of Aeronautics and Mechanical Engineering would allow access to the external network via local PC's, acting as intelligent remote terminals, enabling the preprocessing of input files and pre-analysis of output files.

The main benefits would however be obtained from an educational point of view, since this would allow efficient file and information transfer between PC's, workstations and other computers, allowing several students to analyse individually the same flow simulation output file, either on a graphical station or, if the problem and the hardware allows for it, on a PC.

This should be realized in close association with the future joint CAD/CAE Laboratory of the Departments of Aeronautics and Mechanical Engineering, where 24 Workstations including four color display stations with high resolution, are to be networked.

D. RECOMMENDATIONS AND CONCLUSIONS

The following conclusions and general recommendations can be summarized as outcome of the present investigation.

Discussions have been held with Faculty members and researchers of the Departments of Aeronautics, Mechanical Engineering, Meteorology and Oceonography with regard to their Computational Fluid Dynamics activities, which might have an impact on Propulsion related Research and Education.

More particularly of interest to the present investigation were the engineering needs for CFD and the common interest of Faculty members of the Departments of Aeronautics and Mechanical Engineering for the development and extension of the CFD potential at NPS.

Discussions have been held with the Director of the Computer Center and with members of the Department of Computer Sciences with regard to the present and future possibilities of computer networks, both external and local.

Discussions have been held with researchers and staff at NASA AMES Research Center and at the NEPRF.

1. The advent of high speed computer facilities, coupled with highly efficient algorithms, has rendered the application of Computational Fluid Dynamic simulations of complex flow systems accessible to the integration of advanced design and analysis methodologies.

2. **It is recommended to develop the applications of CFD at the educational level, in the form of visualizations of numerical simulations of basic, flow phenomena. This will also enable the student to investigate, through the graphical analysis of pre-stored computational results on adapted workstations, the influence of geometrical, operational and environmental parameters on the properties and behavior of propulsion systems and their components.**

3. There is a need for the availability of a range of fluid simulation codes ranging from codes with a general application domain for full three dimensional analysis, to codes based on simplified models for more specific applications .

4. **It is recommended to give a strong priority to the available codes**

developed at NASA AMES Research Center, in view of the recent creation of the JOINT INSTITUTE between the Department of Aeronautics and NASA AMES, which provides the framework for the necesary support to the exploitation and eventual extensions or modifications to the computer programs.

5. There is a clear need, expressed at the level of Faculty members of the Aeronautics and Mechanical Enginering Departments for organized support in the use, access and operation of CFD codes, with regard to input and output file management and analysis on high performance engineering and graphical workstations.

6. It is recommended to consider the creation of a support team of software/hardware specialists in order to ensure the continuity of the initial efforts of installation of CFD codes, to guarantee the maintenance and upgrading to new versions or to additional improvements provided by the originators of the codes, to maintain the appropriate documentation, and to provide support to new users.

   The support team should maintain the availability of stored output data files prepared and developed for the support of student courses.

   This last aspect is essential in order to allow the access of the students to the analysis and visualization of flow systems in relation to associated courses .

7. The installation of a CFD support team could be considered in relation with, or as part of, the CAD/CAE laboratory installed jointly by the Departments of Aeronautics and Mechanical Engineering. This seems appropriate since Faculty members of both Departments have expressed a strong interest for the above mentioned organization and management of available, or to be installed, general purpose codes.

   The access of the students to prepared output files for demonstrations, analysis and visualizations could be organized at the CAD/CAE laboratory with the active participation of the support team, or locally within the concerned laboratory when the appropriate equipment is available.

8. The acquisition of advanced graphical stations should be considered in relation with the possibility of running the very performant utility code PLOT3D, developed at NASA AMES, for 3D color graphic flow data representation and visualization.

9. **It is recommended to consider a network of the Personal Computers within the Departements of Aeronautics and Mechanical Engineering to support multiple transfer of files between graphical stations, students and instructors in order to facilitate the above recommendations concerning the introduction of the capabilities and potential of CFD to the students.**

   The development and installation of a PC network on the workstation network could be considered in relation with the department of Computer Sciences.

10. **It is recommended to develop a privileged network connection between the NPS computers and the computer systems at NASA AMES, with dedicated efficient File Transfer Protocols compatible at the two ends.**

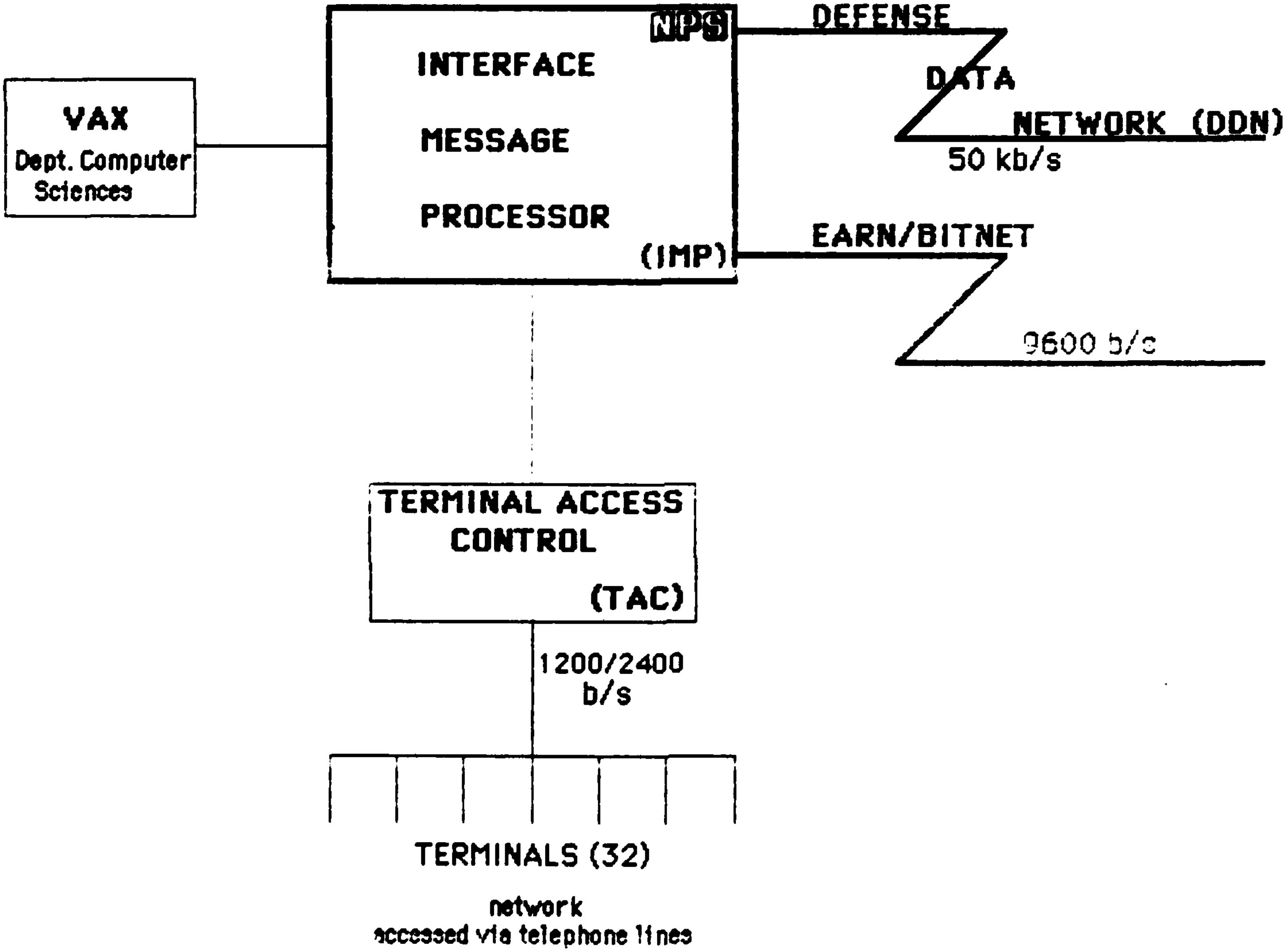


FIGURE 1 : BASIC EXTERNAL NETWORK SYSTEM AT NPS

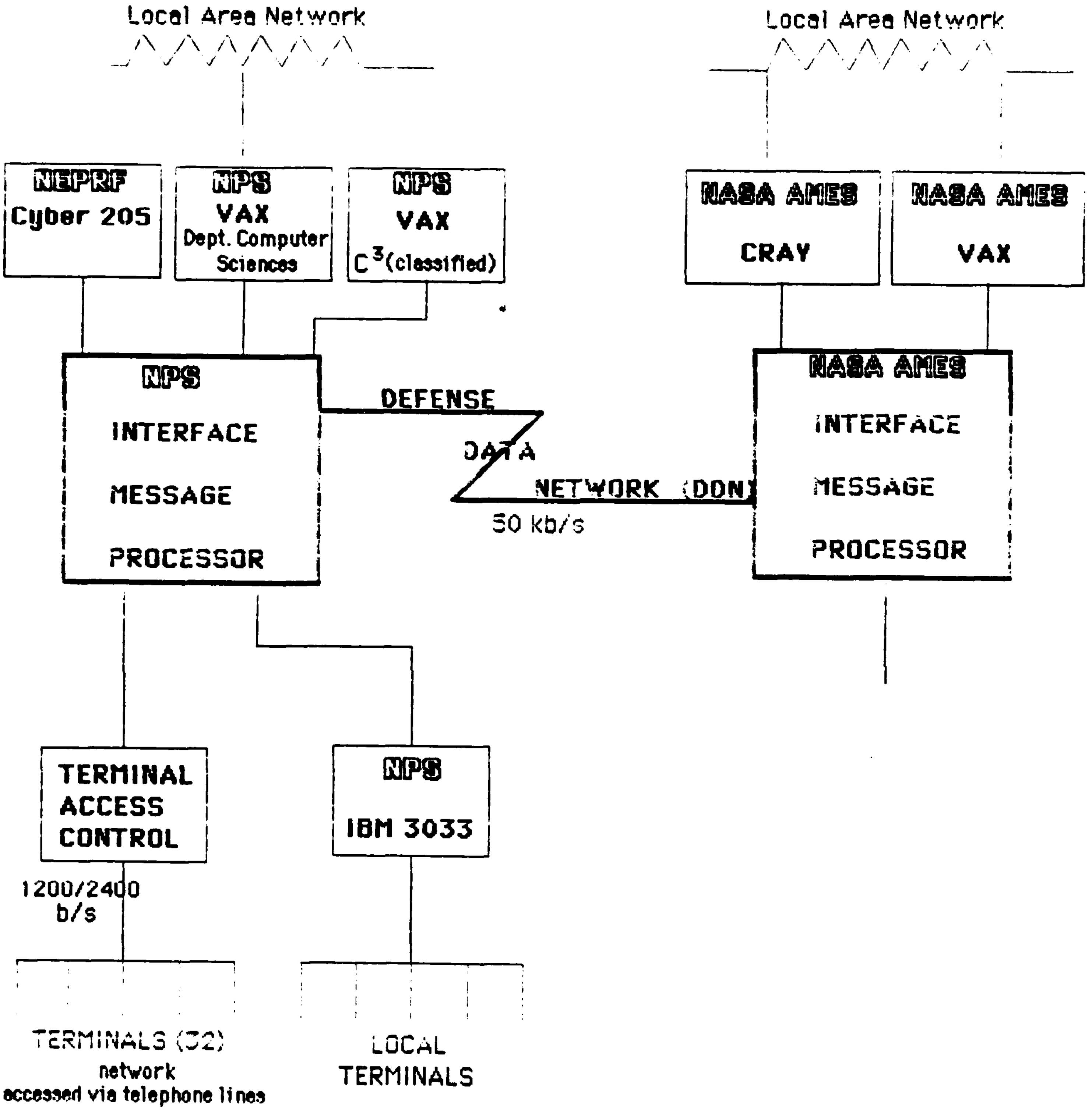


FIGURE 2 · FUTURE NETWORK CONFIGURATION AT NPS

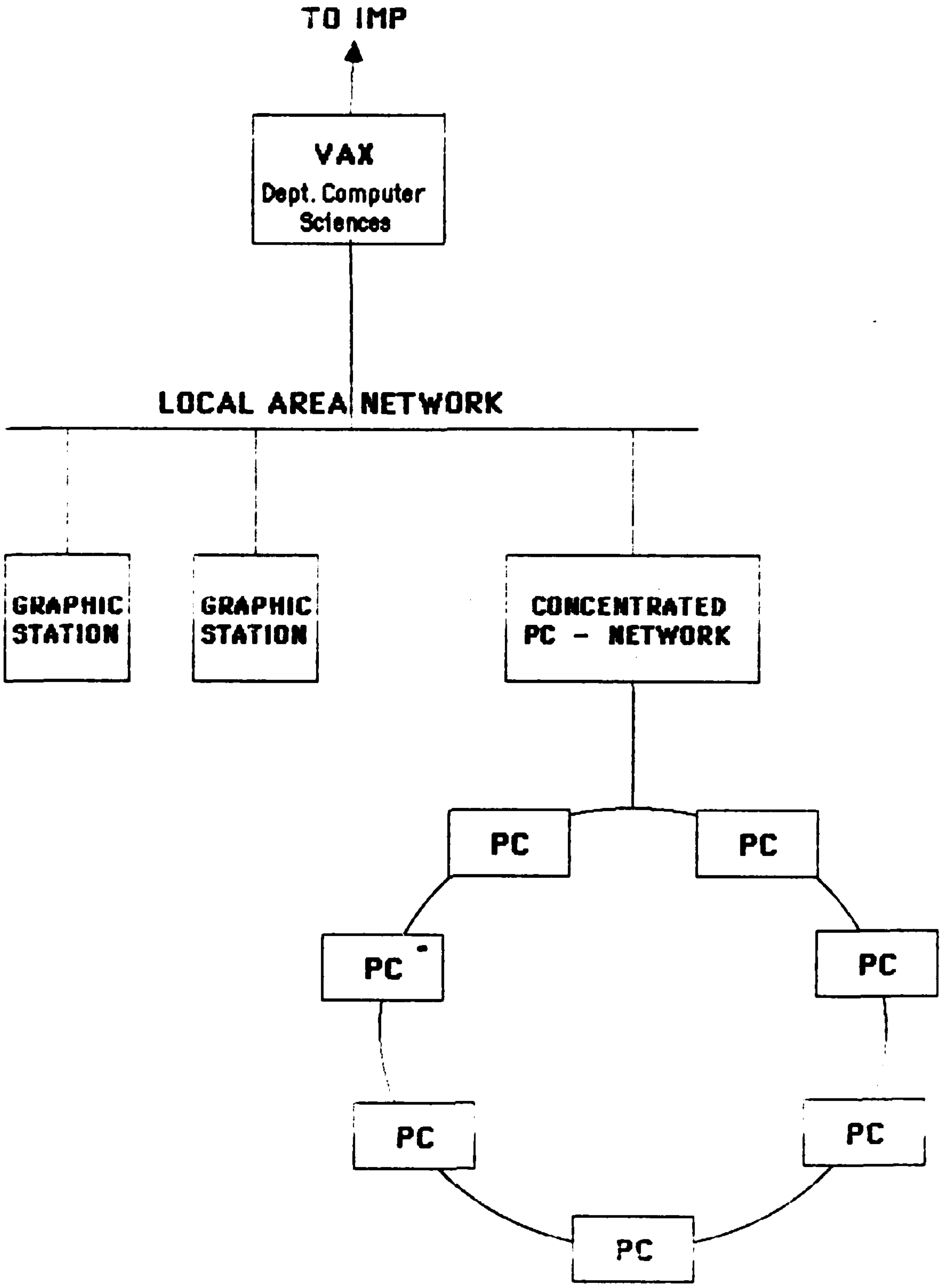


FIGURE 3 : LOCAL AREA NETWORK CONNECTING GRAPHIC STATIONS AND PERSONAL COMPUTERS

APPENDIX A

TECHNICAL DATA SHEETS FOR AERONAUTICAL ORIENTED CFD CODES AVAILABLE AT NPS

CONTENT

A 1

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: BLAYER
NASA Lewis Research Center

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	**Quasi three dimensional**	Two dimensional
Viscous	**Inviscid + boundary layer**	Inviscid
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	**Cascades**	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

Boundary layer analysis with or without separation, applied to cascade flows.

PERSON TO CONTACT:

A. McGuire, Dept. Aeronautics, Turbopropulsion Lab., NPS
N. Sanger, NASA Lewis Research Center

A2

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Q3DFLO-81
Professor Ch. Hirsch
Vrije Universiteit Brussel
Dept. Fluid Mechanics-Brussels 1050 - Belgium

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces:
gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows		**Steady state flows**
Three dimensional	**Quasi three dimensional**		
Two dimensional			
Viscous	**Viscous-Inviscid interaction**		
Inviscid	**Incompressible fluid**	**Perfect gas**	
Real gas			
Subsonic	**Transonic**		Supersonic
Turbulent and laminar	**Turbulent**		Laminar
Internal flow geometry	**Cascades**		External flow
Chemical kinetics	Radiation		Heat transfer

OTHER CHARACTERISTICS

Developed for through-flow and combined blade-to-blade computations in multistage turbomachines of any kind, axial, radial or mixed flow configurations. An option is open for the inviscid interaction of the through-flow with the end-wall boundary layers, calculated with an integral method.

PERSON TO CONTACT:

Professor R. Shreeve, Department Aeronautics, NPS
Professor Ch. Hirsch, VUB, Brussels.

A 3

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE:

EULER 1
Profesor R. Shreeve
Dept. Aeronautics - NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	**Transient flows**	Steady state flows
Three dimensional	Quasi three dimensional	Two dimensional
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

Is presently applicable to the one dimensional Shock-tube problem.

The program is the first step in the development of a general Euler code based on a characteristic equation formulation.

PERSON TO CONTACT:

Professor R. Shreeve, Department Aeronautics, NPS

A 4

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: TSONIC / QSONIC
NASA Lewis Research Center

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	**Quasi three dimensional**	
Two dimensional		
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	**Cascades**	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

These programs solve the subsonic or transonic potential flow through cascades, without shocks, on an axisymmetric blade-to-blade surface of a turbomachine blade row.

The Input/Output files are compatible with an associated through-flow program called MERIDL.

PERSON TO CONTACT:

A. McGuire, Dept. Aeronautics, Turbopropulsion Lab., NPS
N. Sanger, NASA Lewis Research Center

A5

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: MERIDL
NASA Lewis Research Center

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows
Three dimensional
Viscous
Incompressible fluid
Subsonic
Turbulent and laminar
Internal flow geometry
Chemical kinetics

Transient flows
Quasi three dimensional
Inviscid + boundary layer
Perfect gas
Transonic
Turbulent
Cascades
Radiation

Steady state flows
Two dimensional
Inviscid
Real gas
Supersonic
Laminar
External flow
Heat transfer

OTHER CHARACTERISTICS

The program solves the axisymmetric through-flow in ducts with or without blades.

The inlet boundary conditions are imposed along a nearly radial line.

The Input/Output data files are compatible with the blade-to-blade programs TSONIC/QSONIC

PERSON TO CONTACT:

A. McGuire, Dept. Aeronautics, Turbopropulsion Lab., NPS
N. Sanger, NASA Lewis Research Center

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: MESHGEN /TURBO
Professor R. Shreeve
Dept. Aeronautics- NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	**Quasi three dimensional**	Two dimensional
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This program has the same purpose as the through-flow part of Q3DFLO-81, and is based on the same basic approach.

PERSON TO CONTACT:

Professor R. Shreeve, Dept. Aeronautics, NPS

A7

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Supersonic flow in Oscillating Cascades
Professor M. Platzer, Dept. Aeronautics, NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	**Transient flows**	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	Transonic	**Supersonic**
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	**Cascades**	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

The program is based on the method of characteristics and assumes a harmonic time dependence. Its domain of application is oriented towards the flow in oscillating cascades.

PERSON TO CONTACT:

Professor M. Platzer, Dept Aeronautics, NPS

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Unsteady, Inviscid Flow in Cascades
T. Franson, Dept. Aeronautics - NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	**Transient flows**	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	**Cascades**	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

The program treats the unsteady flow through a vibrating cascade and is intended for flutter analysis.

The application of the program is limited to low camber blades, with a thickness/chord ratio not exceeding 10%.

PERSON TO CONTACT:

T. Franson, Dept. Aeronautics, NPS

A9

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Keller Box Method for Boundary Layers
T. Cebeci, Mc Donnell Douglas Corporation and
Dept. Aeronautics, NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + **boundary layer**	Inviscid
Incompressible fluid	Perfect gas	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This program solves the boundary layer equations with a differential method based on the Keller box scheme.

PERSON TO CONTACT:

Professor M. Platzer, Dept. Aeronautics, NPS
Dr. T. Cebeci, Mc Donnell Douglas Corporation and Dept. Aeronautics, NPS

A 10

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Viscid-Inviscid Interaction on Airfoil
T. Cebeci, Mc Donnell Douglas Corporation and
Dept. Aeronautics, NPS

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows		**Steady state flows**
Three dimensional	Quasi three dimensional		**Two dimensional**
Viscous	**Inviscid + boundary layer**		
Inviscid	**Incompressible fluid**	Perfect gas	
Real gas			
Subsonic	Transonic		Supersonic
Turbulent and laminar	Turbulent		Laminar
Internal flow geometry	Cascades		**External flow**
Chemical kinetics	Radiation		Heat transfer

OTHER CHARACTERISTICS

This program is being extended to cover also flow in cascades

PERSON TO CONTACT:

Professor M. Platzer, Dept. Aeronautics, NPS
Dr. T. Cebeci, Mc Donnell Douglas Corporation and Dept. Aeronautics, NPS

A 11

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Flow over Wing-Body Junctions
Boeing Cº, Seattle

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	Quasi three dimensional	Two dimensional
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	Transonic	**Supersonic**
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	**External flow**
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This program calculates the three-dimensional linearized potential flow over a wing-body junction in the subsonic and supersonic ranges. It has no shock capturing capability.

PERSON TO CONTACT:

Professor M. Platzer, Dept Aeronautics, NPS

A 12

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Panel Method
Mc Donnell Douglas Corporation

FLOW MODEL:

Navier-Stokes equations:	full Navier-Stokes thin shear layer model parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
[✓ Linearized potential equation]
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	[✓ Steady state flows]
[✓ Three dimensional]	Quasi three dimensional	[✓ Two dimensional]
Viscous	Inviscid + boundary layer	[✓ Inviscid]
[✓ Incompressible fluid]	Perfect gas	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	[✓ External flow]
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

The program is based on the singularity-panel method and applies to the flow along wings, planforms, jet-flaps.

PERSON TO CONTACT:

Professor M. Platzer, Dept. Aeronautics, NPS

A 13

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: PHOENICS
Cham Ltd, UK

FLOW MODEL:

Navier-Stokes equations:
full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: **gravity**
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	Steady state flows
Three dimensional	Quasi three dimensional	Two dimensional
Viscous	Inviscid + boundary layer	Inviscid
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	Radiation	**Heat transfer**

OTHER CHARACTERISTICS

This is a commercial code, based on the Patankar-Spalding method, and contains a large number of options. Its accuracy is generally limited to first order, which may sometimes call for some caution in the interpretation of results.

PERSON TO CONTACT:

Professor M. Kelleher, Dept. Mechanical Engineering, NPS
Professor D. Salinas, Dept. Mechanical Engineering, NPS

A 14

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Garrett/ AiResearch/NPS Combustion Code
Professor D. Netzer, Dept. Aeronautics, NPS

FLOW MODEL:

Navier-Stokes equations: **full Navier-Stokes**
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	Quasi three dimensional	Two dimensional
Viscous	Inviscid + boundary layer	Inviscid
Incompressible fluid	Perfect gas	**Real gas**
Subsonic	Transonic	Supersonic
Turbulent and laminar	**Turbulent**	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	**Radiation**	**Heat transfer**

OTHER CHARACTERISTICS

This code is an extension of the original Garrett/AiResearch combustion code to include additional geometrical configurations, a fuel wall and modified fuel properties to reflect the properties of Plexiglass. The modifications are directed towards solid fuel ramjet combustion processes.

PERSON TO CONTACT:

Professor D. Netzer, Dept. Aeronautics, NPS

A 15

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: UNDSAFE
Professor K.T. Yang, University of Notre-Dame

FLOW MODEL:

Navier-Stokes equations: **full Navier-Stokes**
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: **gravity**
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	Inviscid
Incompressible fluid	Perfect gas	**Real gas**
Subsonic	Transonic	Hypersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	External flow
Chemical kinetics	**Radiation**	**Heat transfer**

OTHER CHARACTERISTICS

This code is intended for fire modeling computations in enclosures, in particular distribution and spreading of smoke in aircraft cabins and submarines.

Extensions are considered to incorporate chemical reactions and complex geometries.

PERSON TO CONTACT:

Professor K. T. Yang, University of Notre-Dame, Dept. Aerospace and Mechanical Engineering Professor M. Kelleher, Dept. Mechanical Engineering, NPS

APPENDIX B

SELECTION OF AERONAUTICAL ORIENTED CFD CODES AVAILABLE FROM COSMIC

AERODYNAMICS

Collection

a selection of
computer program abstracts

COSMIC
NASA's Computer Software Management and Information Center
112 Barrow Hall • The University of Georgia • Athens, GA 30602 • USA
Phone: (404) 542-3265 Telex: 810-754-3908

A Selection of

AERODYNAMICS

Computer Programs

COSMIC - the Computer Software Management and Information Center - has operated since 1966 under contract to the National Aeronautics and Space Administration (NASA). COSMIC is the center for collection, evaluation, and dissemination of computer software developed by NASA and NASA contractors.

Over 1100 computer programs are currently in the COSMIC inventory and available to business, industry, universities, and other government agencies. Programs cover the many areas of NASA project involvement, ranging from structural mechanics and thermodynamics to data management and system development.

COSMIC offers a number of programs in this general area of interest. The enclosed abstracts describe some of these programs to give you an idea of the variety we offer. If you do not find a program that meets your needs, call our Customer Service staff. They'll provide a more detailed search, at no charge, to determine whether we have programs for your specific applications. Phone: (404) 542-3265

NASA

National Aeronautics and
Space Administration

COSMIC
Computer Services Annex
The University of Georgia
Athens, GA 30602

AERODYNAMICS

1. AEROX- TRANSONIC AIRCRAFT AERODYNAMICS TO HIGH ANGLES OF ATTACK

2. GRAPE- TWO-DIMENSIONAL GRIDS ABOUT AIRFOILS AND OTHER SHAPES BY THE USE OF POISSON'S EQUATION

3. PANAIR- PREDICTING SUBSONIC OR SUPERSONIC LINEAR POTENTIAL FLOWS ABOUT ARBITRARY CONFIGURATIONS USING A HIGHER ORDER PANEL METHOD

4. TAIR- TRANSONIC AIRFOIL ANALYSIS COMPUTER CODE

5. AERODYNAMIC ANALYSIS OF WING-BODY-TAIL CONFIGURATIONS IN SUBSONIC AND SUPERSONIC FLOW

6. SUPER/HYPERSONIC INVISCID FLOW AROUND REAL CONFIGURATIONS

7. APAS- AERODYNAMIC PRELIMINARY ANALYSIS SYSTEM

8. FLO 22- TRANSONIC FLOW PAST A SWEPT WING

9. NASA- LOCKHEED MULTI-ELEMENT AIRFOIL ANALYSIS

10. RAXBOD- INVISCID TRANSONIC FLOW OVER AXISYMMETRIC BODIES

11. THREE DIMENSIONAL POTENTIAL FLOW PROGRAM WITH GEOMETRY PACKAGE FOR GENERATION OF INPUT DATA

12. WIBCO- WING BODY CODE FOR TRANSONIC FLOW FIELD ANALYSIS OF WING-FUSELAGE CONFIGURATION

13. PROFILE- THE EPPLER PROGRAM FOR THE DESIGN AND ANALYSIS OF LOW-SPEED AIRFOILS

14. AERODYNAMIC DESIGN AND ANALYSIS OF SUPERSONIC AIRCRAFT

15. THREE DIMENSIONAL NAVIER-STOKES EQUATIONS FOR FLOW OVER NONAXISYMMETRIC NOZZLE CONFIGURATIONS

16. VLM- THE EXTENDED NASA LANGLEY VORTEX LATTICE METHOD

17. AFSMO/AFSCL- AIRFOIL SMOOTHING AND SCALING

18. WAVDRAG- ZERO-LIFT WAVE DRAG OF COMPLEX AIRCRAFT CONFIGURATIONS

19. COREL/W12SC3- PROGRAMS FOR SUPERSONIC WING DESIGN AND ANALYSIS

20. NASCRIN- NUMERICAL ANALYSIS OF SCRAMJET INLET

21. SOPA- SECOND ORDER POTENTIAL ANALYSIS AND OPTIMIZATION

22. WINGDES- DESIGN OF WING SURFACES AT SUBSONIC OR SUPERSONIC SPEEDS

23. NCOREL- NONCONICAL RELAXATION FOR THREE-DIMENSIONAL NONLINEAR SUPERSONIC POTENTIAL FLOW

24. AIRFOIL- AERODYNAMIC CHARACTERISTICS OF NACA 16-SERIES AIRFOILS

25. SIMP- SUPERSONIC IMPLICIT MARCHING PROGRAM FOR NONLINEAR FULL POTENTIAL ANALYSIS

26. VORSTAB- A COMPUTER PROGRAM FOR CALCULATING LATERAL-DIRECTIONAL STABILITY DERIVATIVES WITH VORTEX FLOW EFFECT

*NOTE:This is only a selection of the programs COSMIC offers in this area. For information on other programs, contact COSMIC's Customer Support Staff.

APPENDIX C

TECHNICAL DATA SHEETS FOR AERONAUTICAL ORIENTED CFD CODES AVAILABLE AT OTHER INSTITUTIONS

CONTENT

C1

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: ARC2D/ARC3D
NASA AMES Research Center

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	**Supersonic**
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	**External flow**
Chemical kinetics	Radiation	**Heat transfer**

OTHER CHARACTERISTICS

This is a general, highly tested and accurate code for two and three dimensional viscous and inviscid external flows, covering the range of subsonic, transonic and supersonic flows, both in the steady and unsteady modes.* Its present geometrical domain of application covers single airfoils.

PERSON TO CONTACT:

Dr. T. Holst, Chief of Applied Computational Fluids Branch, NASA AMES Research Center
Dr. T. Pulliam, Computational Fluid Dynamics Branch, NASA AMES Research Center

C2

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: TAIR
NASA AMES Research Center

FLOW MODEL:

Navier-Stokes equations:	full Navier-Stokes thin shear layer model parabolized Navier-Stokes
Viscous-Inviscid interaction model	
Distributed loss model	
Boundary layer equations	
Euler equations	
Potential equation	
Linearized potential equation	
External forces:	gravity buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible Fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	**Supersonic**
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	**External Flow**
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This code is based on the full potential equation in conservative form and applies to three dimensional wings and wing-body junctions.

PERSON TO CONTACT:

Dr. T. Holst, Chief of Applied Computational Fluids Branch, NASA AMES Research Center

C3

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: TURF
NASA AMES Research Center

FLOW MODEL:

Navier-Stokes equations: **full Navier-Stokes**
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	**Supersonic**
Turbulent and laminar	**Turbulent**	Laminar
Internal flow geometry	Cascades	**External flow**
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

The code treats also axisymmetric configurations and has some interesting features from user's point of view. In particular, it has a build-in mesh generation system and plotting routines for output display.

An interesting aspect is the option to compare different turbulence models, from zero equation to different two equation models

PERSON TO CONTACT:

Dr. T. Holst, Chief of Applied Computational Fluids Branch, NASA AMES Research Center
Dr.T.Coackley, Applied Fluid Dynamics Branch, NASA AMES Research Center

C4

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: INS3D
NASA AMES Research Center

FLOW MODEL:

Navier-Stokes equations: **full Navier-Stokes**
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	Transient flows	**Steady state flows**
Three dimensional	Quasi three dimensional	Two dimensional
Viscous	*Inviscid + boundary layer*	Inviscid
Incompressible fluid	Perfect gas	Real gas
Subsonic	Transonic	Supersonic
Turbulent and laminar	Turbulent	Laminar
Internal flow geometry	Cascades	**External flow**
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This incompressible Navier-Stokes code is based on the Pseudo-compressibility approach for the treatment of the incompressibility condition. This results in an efficient algorithm for stationary flows at the cost of preventing time accurate simulations.

The code allows the easy implementation of various turbulence models and can be applied to internal as well as external flows.

PERSON TO CONTACT:

Dr. T. Holst, Chief of Applied Computational Fluids Branch, NASA AMES Research Center
Dr. D. Kwak, Applied Computational Fluids Branch, NASA AMES Research Center

C5

TECHNICAL DESCRIPTION SHEET

NAME AND ORIGIN OF CODE: Stator-Rotor Interaction Code
NASA AMES Research Center

FLOW MODEL:

Navier-Stokes equations: full Navier-Stokes
thin shear layer model
parabolized Navier-Stokes

Viscous-Inviscid interaction model
Distributed loss model
Boundary layer equations
Euler equations
Potential equation
Linearized potential equation
External forces: gravity
buoyancy

DOMAIN OF APPLICATION

Transient + steady flows	**Transient flows**	Steady state flows
Three dimensional	Quasi three dimensional	**Two dimensional**
Viscous	Inviscid + boundary layer	**Inviscid**
Incompressible fluid	**Perfect gas**	Real gas
Subsonic	**Transonic**	**Supersonic**
Turbulent and laminar	**Turbulent**	Laminar
Internal flow geometry	**Cascades**	External flow
Chemical kinetics	Radiation	Heat transfer

OTHER CHARACTERISTICS

This code computes the interacting flow through the stator and the rotor of a two-dimensional cascade, with the restriction of an equal number of blades in both cascades. Due to the considerable amount of computations involved, this code might be expensive to apply.

A movy of the time evolution of the computed results is available at the Turbopropulsion Laboratory at NPS.

PERSON TO CONTACT:

Dr. T. Holst, Chief of Applied Computational Fluids Branch, NASA AMES Research Center
Dr. M.M.Rai, Informatics General Corp., Palo Alto, California

Distribution List

1.	Naval Air Systems Command	
	Washington, DC 20361	
	Attention: Code AIR 93D	1
	Code AIR 931E	1
2.	Office of Naval Research	
	800 N. Quincy Street	
	Arlington, VA 22217	
	Attention: Dr. R. Whitehead	1
	Dr. A. D. Wood	1
	Dr. F. Hansen	1
3.	NASA Ames Research Center	
	Moffett Field, CA 94035	
	Attn: Dr. T. Holst	1
	Dr. M. M. Rai	1
	Dr. D. Kwak	1
	Dr. T. Coakley	1
	Dr. T. Pulliam	1
4.	NASA Lewis Research Center	
	21000 Brookpark Rd.	
	Cleveland, OH 44135	
	Attn: N. Sanger (MS. 5-11)	1
5.	Naval Postgraduate School	
	Monterey, CA 93943-5000	
	Attn: Library	2
	Code 06	1
	Code 012	1
	Code 67	15
	Code 67Am, 67Fr, 67Nt, 67Pl, 67Sf	5
	Code 69	15
	Code 69Kr, 69Sa	2
6.	Prof. Dr. Jr. Ch. Hirsch	1
	VRIJE Universiteit Brussel	
	Pleinlann 2,	
	1050 Brussel, Belgium	
7.	Defense Technical Information Center	
	Cameron Station	2
	Arlington, VA 22341	

END
DATE
FILMED
JAN
1988