Computational Fluid Dynamics at the Lewis Research Center
An Overview

By
Robert M. Stubbs
NASA-Lewis Research Center
Cleveland, Ohio 44135

Lewis is a multidisciplinary Center with strong research and development programs in aeronautical and space propulsion, power, space communications, space experiments and materials. Computational fluid dynamics (CFD) is playing an important and growing role in most of these areas. This presentation describes how CFD is integrated into these programs and highlights elements of the CFD activities. Examples are presented of codes developed to predict flow fields in advanced propulsion systems and several of the code validation experiments are described. As will be evident in the several Lewis authored papers to be presented at this conference, the CFD effort at Lewis ranges from basic research on new and improved algorithms through code development to the application of these codes to specific engineering problems. Because of the substantial improvement in CFD's predictive capability its use at Lewis is on a steep growth path, spreading rapidly into new areas which had not traditionally taken advantage of the techniques of numerical simulation. The presentation concludes with a discussion of multidisciplinary codes and the future direction of CFD at Lewis.

LEWIS COMPUTER RESOURCES

- CRAY X-MP/24
- SCIENTIFIC VAX CLUSTER
- 2 AMDAHLS (VM/CMS AND MVS/XA)
- ALLIANT FX/8 PARALLEL PROCESSOR
- ADVANCED WORK STATIONS (SILICON GRAPHICS, SUN, APOLL0)
- T1 LINK TO NAS

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION

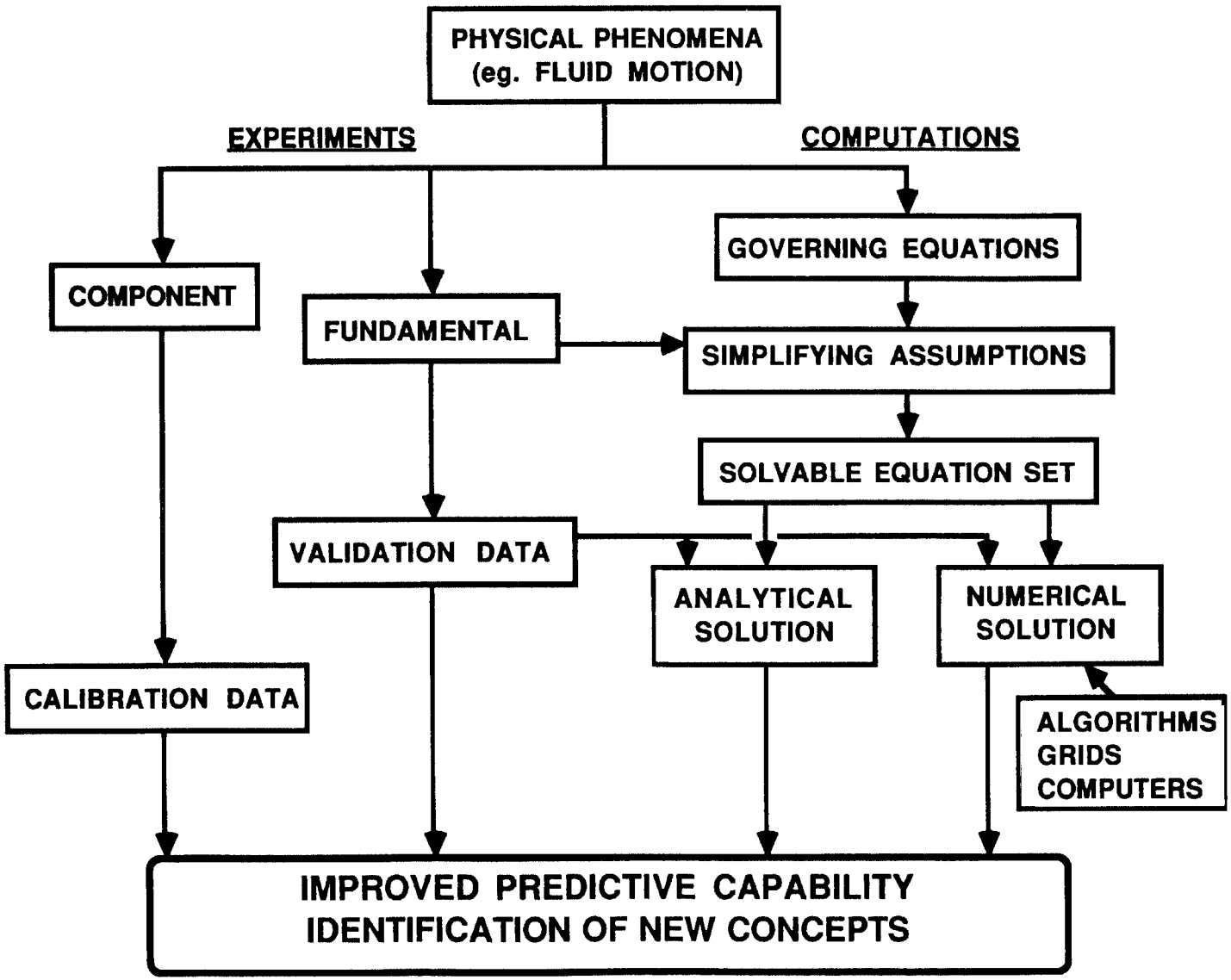
OBJECTIVE

- TO DEVELOP ADVANCED COMPUTATIONAL METHODS REQUIRED FOR THE SOLUTION OF INTERNAL FLUID MECHANICS AND STRUCTURAL PROBLEMS

ROLE

- GENERATE NEW IDEAS/APPROACHES TO PROPULSION RESEARCH THROUGH INTERACTION WITH LeRC STAFF
- PROVIDE OPPORTUNITY TO ICOMP PERSONNEL TO DO ORIGINAL RESEARCH UTILIZING WORLD-CLASS COMPUTERS

CFD'S ROLE IN THE PROBLEM SOLVING PROCESS

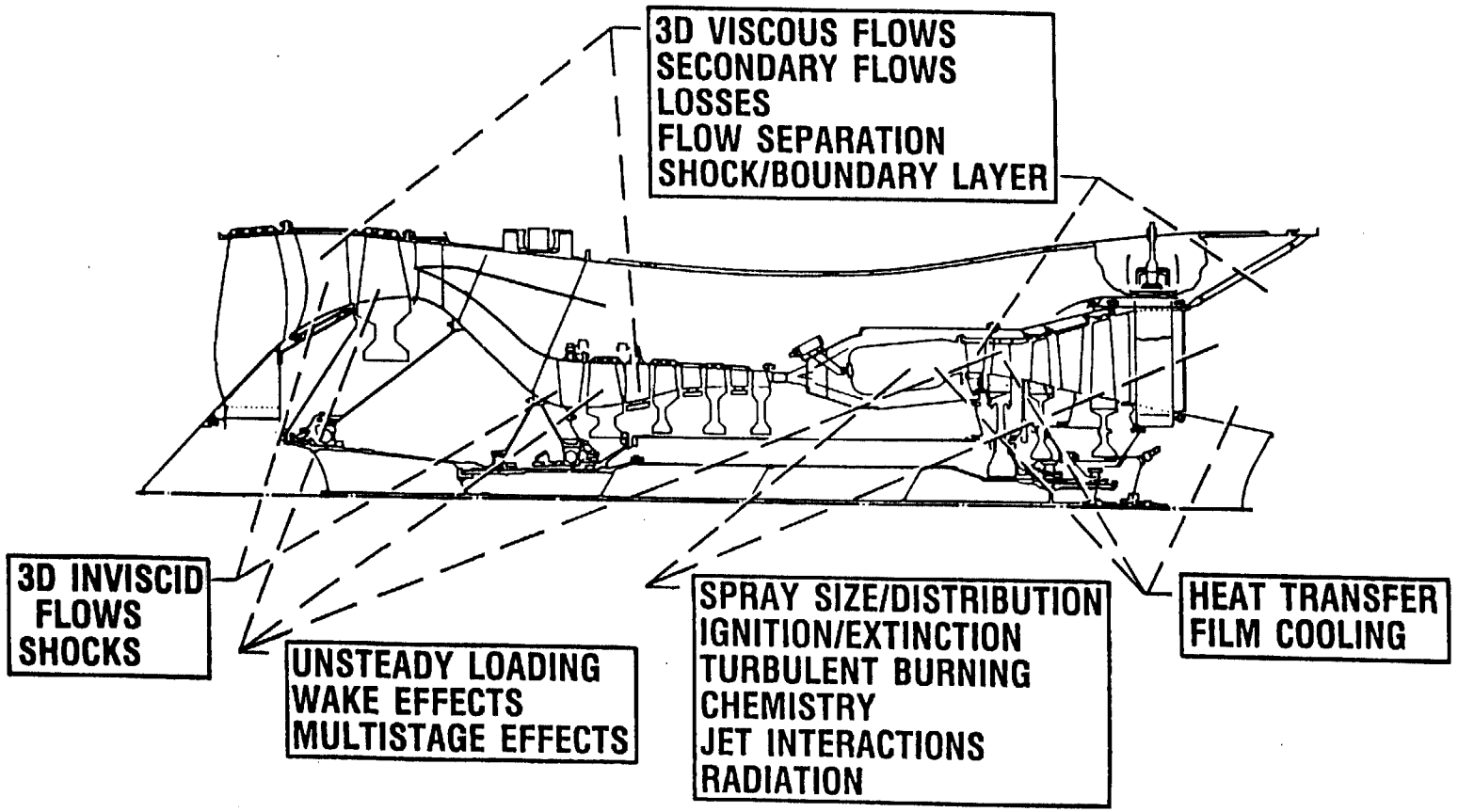


COMPUTATIONAL FLUID DYNAMICS AT LEWIS

WHERE DOES CFD PLAY A ROLE?

- **AERO PROPULSION**
- **SPACE PROPULSION**
- **OTHER**
  - **SPACE POWER**
  - **MATERIALS PROCESSING**
  - **FLUIDS IN MICROGRAVITY**

CONCEPTUAL ADVANCED TURBOFAN ENGINE PHYSICAL PHENOMENA REQUIRING ANALYSES



RVC3D (ROTOR VISCOUS CODE 3-D)

BY DR. R. V. CHIMA

- NAVIER-STOKES ANALYSIS FOR 3D FLOWS IN TURBOMACHINERY
- STACKED C-TYPE GRIDS FOR AXIAL OR CENTRIFUGAL MACHINES
- THIN-LAYER NAVIER-STOKES FORMULATION RETAINS HUB-TO-TIP AND BLADE-TO-BLADE VISCOUS TERMS
- BALDWIN-LOMAX TURBULENCE MODEL
- EXPLICIT 4-STAGE RUNGE-KUTTA TIME-MARCHING SCHEME WITH VARIABLE TIME STEP AND IMPLICIT RESIDUAL SMOOTHING
- HIGHLY VECTORIZED FOR CRAY X-MP

RPLUS 3D

BY DRS. S.T. YU, J. S. SHUEN & P. TSAI

- 3D NAVIER-STOKES CODE FOR CHEMICALLY REACTING FLOWS

- FINITE RATE CHEMISTRY MODEL
  - O 9 SPECIES INVOLVING O, H, AND N
  - O 18 REACTION STEPS

- TIME-MARCHING LU SCHEME

- FAST AND ROBUST

PROTEUS

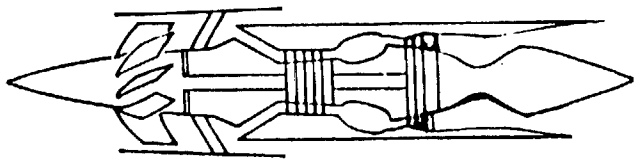
- IMPLICIT 2D AND 3D NAVIER-STOKES CODE
- READILY MODIFIED, WELL DOCUMENTED GENERAL SOLVER
- FULL, COMPRESSIBLE NAVIER-STOKES WITH ENERGY EQUATION
- EULER AND THIN-LAYER OPTIONS
- LINEARIZED BLOCK IMPLICIT (LBI) SOLVER
- IMPLICIT BOUNDARY CONDITIONS
- FIRST OR SECOND ORDER IN TIME
- ALGEBRAIC (BALDWIN-LOMAX) AND 2-EQUATIONTURBULENCE MODELS CURRENTLY BEING IMPLEMENTED

MULTISTAGE AVERAGE PASSAGE CODE

BY DR. JOHN ADAMCZYK

- MULTISTAGE SIMULATIONS FOR ARBITRARY GEOMETRIES
- 3D VISCOUS
- EXPLOITS MACROTASKING
- FIRST SIMULATION OF A 3D MULTISTAGE TURBINE
- COMPATIBLE WITH THE DESIGN ENVIRONMENT
- CODE/METHODOLOGY CURRENTLY BEING USED IN DESIGN OF TURBOMACHINERY

PUTTING IT ALL TOGETHER



INLETS, DUCTS, AND NOZZLES

TURBOMACHINERY

CHEMICAL REACTING FLOWS

COMPUTATIONAL AND EXPERIMENTAL TECHNOLOGY

INTEGRATED MULTIDISCIPLINARY ANALYSIS AND TEST

NUMERICAL PROPULSION SYSTEM SIMULATION

LEWIS RESEARCH CENTER

NUMERICAL SIMULATION OF NONLINEAR DEVELOPMENT OF INSTABILITY WAVES	R. MANKBADI
TIME DEPENDENT VISCOUS INCOMPRESSIBLE NAVIER-STOKES EQUATIONS	J. GOODRICH
PROGRESS TOWARD THE DEVELOPMENT OF AN AIRFOIL ICING ANALYSIS CAPABILITY	M. POTAPCZUK C. BIDWELL B. BERKOWITZ
THE BREAKUP OF TRAILING-LINE VORTICES	D. JACQMIN
THREE DIMENSIONAL SIMULATION OF SUPERSONIC REACTING FLOWS WITH FINITE RATE CHEMISTRY	S. YU J. SHUEN P. TSAI
SIMULATION OF TURBOMACHINERY FLOWS	J. ADAMCZYK
AUTOMATED DESIGN OF CONTROLLED DIFFUSION BLADES	J. SANZ
NUMERICAL ANALYSIS OF FLOW THROUGH OSCILLATING CASCADE SECTIONS	D. HUFF
NUMERICAL ANALYSIS OF THREE-DIMENSIONAL VISCOUS INTERNAL FLOWS	R. CHIMA J. YOKOTA
A NUMERICAL STUDY OF THE HOT GAS ENVIRONMENT AROUND A STOVL AIRCRAFT IN GROUND PROXIMITY	T. VAN OVERBEKE J. HOLDEMAN
CFD ANALYSIS FOR HIGH SPEED INLETS	T. BENSON
FLUX SPLITTING ALGORITHMS FOR TWO-DIMENSIONAL REAL GAS FLOWS	J. SHUEN M. LIOU

FUTURE DIRECTIONS

- SEVERAL FACTORS CONTRIBUTING TO A CFD EXPLOSION AT LEWIS
  - IMPROVED PREDICTIVE CAPABILITY
  - INCREASE IN MACHINE CAPACITY AND AVAILABILITY

- CFD NO LONGER LOCALIZED, NOW IN WIDE USE IN SEVERAL DISCIPLINE AREAS

- GROWING EMPHASIS ON MULTIDISCIPLINARY SYSTEM SIMULATIONS

SESSION II

CENTER OVERVIEWS

(Continued)

Chairman:
Paul Kutler
Chief, Fluid Dynamics Division
NASA Ames Research Center