DTIC FILE COPY

(1)

AGARD-AR-257

AGARD-AR-257

AD-A211 102

AGARD

ADVISORY GROUP FOR AEROSPACE RESEARCH & DEVELOPMENT

7 RUE ANCELLE 92200 NEUILLY SUR SEINE FRANCE

AGARD Advisory Report No.257

Technical Evaluation Report on the Fluid Dynamics Panel Symposium on Validation of Computational Fluid Dynamics

DTIC ELECTE JUN 22 1989 H

NORTH ATLANTIC TREATY ORGANIZATION

DISTRIBUTION AND AVAILABILITY ON BACK COVER

DISTRIBUTION STATEMENT A

AGARD-AR-257

NORTH ATLANTIC TREATY ORGANIZATION

ADVISORY GROUP FOR AEROSPACE RESEARCH AND DEVELOPMENT

(ORGANISATION DU TRAITE DE L'ATLANTIQUE NORD)

AGARD Advisory Report No.257

TECHNICAL EVALUATION REPORT
on the
FLUID DYNAMICS PANEL SYMPOSIUM
on
VALIDATION OF COMPUTATIONAL FLUID DYNAMICS

by

P.W.Sacher
Messerschmitt-Bölkow-Blohm-GmbH
FE11
Postfach 80 11 60
D-8000 München 80
Federal Republic of Germany

Edited by

R.G.Bradley, Jr
Director, Aerospace Technology Dept.
GD/FW-MZ 2888
Fort Worth Division
P.O.Box 748
Fort Worth TX 76101
USA

W.Schmidt
Deputy Director
Dornier 328 Program
Dornier GmbH, EY
P.O.Box 1420
D-7990 Friedrichshafen 1
Federal Republic of Germany

This Advisory Report was produced at the request of the Fluid Dynamics Panel of AGARD

THE MISSION OF AGARD

According to its Charter, the mission of AGARD is to bring together the leading personalities of the NATO nations in the fields of science and technology relating to aerospace for the following purposes:

— Recommending effective ways for the member nations to use their research and development capabilities for the common benefit of the NATO community;

— Providing scientific and technical advice and assistance to the Military Committee in the field of aerospace research and development (with particular regard to its military application);

— Continuously stimulating advances in the aerospace sciences relevant to strengthening the common defence posture;

— Improving the co-operation among member nations in aerospace research and development;

— Exchange of scientific and technical information;

— Providing assistance to member nations for the purpose of increasing their scientific and technical potential;

— Rendering scientific and technical assistance, as requested, to other NATO bodies and to member nations in connection with research and development problems in the aerospace field.

The highest authority within AGARD is the National Delegates Board consisting of officially appointed senior representatives from each member nation. The mission of AGARD is carried out through the Panels which are composed of experts appointed by the National Delegates, the Consultant and Exchange Programme and the Aerospace Applications Studies Programme. The results of AGARD work are reported to the member nations and the NATO Authorities through the AGARD series of publications of which this is one.

Participation in AGARD activities is by invitation only and is normally limited to citizens of the NATO nations.

The content of this publication has been reproduced
directly from material supplied by AGARD or the author.

Published May 1989

Copyright © AGARD 1989
All Rights Reserved

ISBN 92-835-0504-2

Printed by Specialised Printing Services Limited
40 Chigwell Lane, Loughton, Essex IG10 3TZ

FOREWORD

This report reviews and evaluates the AGARD Fluid Dynamics Panel Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes and to ensure that the mathematical and numerical schemes employed in the codes correctly model the critical physics of the flow field under consideration. The evaluator addresses separately each of the papers presented and makes general comments on the seven major topic sessions. In addition, a Poster Presentation was reviewed in detail.

The author summarises the papers, makes comments on the programme and overall outcome of the Symposium and tries to identify gaps where future work is needed. It was evident that the new possibilities of CFD provide efficient tools for analysis and design in the aeronautical industry, but it was also evident that in spite of the existence of a number of excellent experimental databases there is still a need for efforts in validating the computer programs by experiment as well as by numerical exercises. Nevertheless CFD is even today playing a more and more important supplementary role to experimental flow simulation.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP437 Volume I and are listed in an Appendix to this report. The Poster Papers, also listed in an Appendix, are published in CP437 Volume II.

* * *

Le présent rapport fait le point et donne une appréciation du Symposium intitulé "La validation du Calcul en Dynamique des Fluides", organisé par le Panel AGARD de la Dynamique des Fluides à Lisbon au Portugal le 2-5 mai 1988. Le Symposium avait pour objet de définir l'état de l'art dans le domaine de la validation des codes machine et de s'assurer que les schémas mathématiques et numériques utilisés dans ces codes permettent la modélisation fidèle des critères physiques du champ d'écoulement considéré.

Le rapporteur évalue chaque présentation et fait des commentaires d'ordre général des sept réunions principales. En outre, le rapport comporte un examen détaillé d'une séance d'information/exposition. L'Auteur donne un résumé de chaque présentation, commente le programme ainsi que les résultats globaux du symposium et signale les besoins en matière du futurs travaux.

S'il semble évident que les nouvelles possibilités offertes par le CDF représentent des outils performants d'analyse et de conception pour l'industrie aéronautique, il est également évident que malgré l'existence d'un certain nombre d'excellentes bases de données expérimentales, des efforts restent à faire en ce qui concerne la validation des programmes informatiques tant au moyen de méthodes expérimentales que par des exercices numériques,

Ceci étant dit, le CDF joue à l'heure actuelle un rôle auxiliaire de plus en plus important dans la simulation expérimentale des écoulements. Les communications présentées lors du symposium sont publiées dans le Compte rendu de Conférence AGARD CP 437 Volume I dont une liste en est donnée à l'annexe au présent rapport.

Une liste des réunions d'information/exposition est donnée en annexe au présent rapport et les communications sont publiées au document AGARD CP 437 Volume II.

Accession For
NTIS GRA&I ☑
DTIC TAB ☐
Unannounced ☐
Justification
By
Distribution/
Availability Codes
Avail and/or
Dist Special
A-1

CONTENTS

62.th MEETING OF THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS : TECHNICAL EVALUATION REPORT

by

P.W.Sacher
Advanced Design Department FE1
Military Aircraft Division
Messerschmitt-Bölkow-Blohm GmbH
Postfach 8o116o
D-8000 München 80 - FRG

S U M M A R Y

"VALIDATION OF COMPUTATIONAL FLIGHT DYNAMICS" was the subject of the 62.nd Meeting of the Fluid Dynamics Panel of AGARD, held in Lisbon, Portugal, in May 1988. 37 Papers have been presented in 7 sessions. The Call-for-Papers was focused on a compilation of the State of Art of Validation of computer codes, "...to insure, that the mathematical and numerical schemes employed in the code model correctly the critical physics of the flow field". This ambitious aim was reached only to a degree by comparison of experimental and computational data. But at least some of the papers have attempted to validate computer codes by the use of recently developed "Data Bases" providing also detailed information on 3D-flow field characteristics. The reviewer summarizes the papers, comments on the overall program, and tries to identify existent gaps for future work.

1. INTRODUCTION

The 62.th Meeting of the Fluid Dynamics Panel was held from the 2nd to the 5th of May, 1988 in Lisbon, Portugal, at the facilities of the National Civil Engineering Laboratory. The Call for Papers was focussed on "Validation of Computational Fluid Dynamics", the program committee had developed the following general objectives :

> "The Aerospace Industry has come to realize that Computational Fluid Dynamics offers great potential as an analysis and design tool. CFD methods of today can simulate flows about complex geometries with simplified flow physics or flows about simple geometries with complex flow physics. Significant progress has been made in computational hardware, solution algorithms, physical modeling, and grid generation.
> The **ultimate goal of CFD development** is a fully mature design and analysis capability, that is user friendly, cost effective, numerical accurate and fully verified by detailed experimental comparisons.CFD code validation is accomplished by detailed flow field comparison of numerical solutions with experimental data to insure that the mathematical and numerical schemes employed in the code accurately model the critical physics of the flow field.
> The symposium will **concentrate on the validation of computational methods**. The objectives are to identify the level of agreement of physical models and numerical solution algorithms with experimental data; to identify regions of validity for given flow solvers; and to identify flow regimes where significant gaps exist and further work is warranted.
> *Three following areas are to be highlighted especially :*
>
> - CFD Solution Evaluations
> - Validation Experiments
> - Comparisons of CFD Solutions with Experiment."

2. GENERAL COMMENTS

According to the professional experience of the AGARD headquarters staff personal and to the efforts of the local host and the national Portuguese delegation to AGARD, the meeting was excellent organized. The programme of the symposium was one of the most comprehensive ones, carried out by the Fluid Dynamics Panel within the past years. More than 90 papers had been submitted to the program committee. So the Call for Papers was really successful and sofar the timing of the meeting appropriate. 172 attendees had been registrated and took actively part in the discussions following the presentations. More than other symposia, organized by AGARD during the past, this meeting was attractive to the **experimentalists and to the theoreticians**, for the engineer working on technical applications and for those working in research institutes. So people from different technical disciplines were mixed during the week of this symposium - following the "Spirit of AGARD".

The Fluid Dynamic Panel has a traditional background in the theme of validation of computer codes :

- o 1976 - 1979 Working Group 04 :
  "Experimental data base for computer program assessment".
  (AGARD-AR-138)
- o 1981 - 1984 Working Group 06 :
  "Large scale computing in aeronautics".
  (AGARD-AR-209)
- o 1982 - 1985 Working Group 07 :
  "Testcases for inviscid flow field methods".
  (AGARD-AR-211)
- o 1982 - 1986 Working Group 08 :
  "Aerodynamics of aircraft afterbody".
  (AGARD-AR-226)
- o 1987 - Working Group 13 :
  "Air intakes for high speed vehicles".
  (to be published)

The symposium lasted three and a half days, the program included 39 papers (37 have been presented) and a Poster Session displaying 16 additional contributions. The program committee had organized seven major sessions and a round table discussion at the end of the meeting:

Session I	- CFD Validation Concepts	
Session II	- External Flow I	: Airfoils
Session III	- External Flow II	: Vortex flows
Session IV	- External Flow III	: Wings/wing body
Session V	- External Flow IV	: High speed flows
	- Poster Paper Session	
Session VI	- Internal Flow I	: Turbomachinery
Session VII	- Internal Flow II	: Intakes and ducts

The full program is given in Appendix B, the papers #13 and #39 and the contributions #8,#9 and #10 to the Poster Session of the written program had been withdrawn. **Session I** presented three (invited) papers, covering validation philosophy, benchmark experiments and accuracy assessment. **Session II** had five papers dealing with 2D flows, four of them reporting on Navier-Stokes solutions. **Session III** covered the region of 3D separated Vortex Flows based on experimental work and solutions of the Euler equations with four papers. **Session IV** showed applications on wings and wing/bodies, seven papers were presented, most of them reporting on viscous flow solutions. **Session V** on High speed flows was relativly short with five papers, but there were good examples for the objectives of the meeting. **Session VI** consisted of seven papers, the first giving an excellent introduction to internal flow problems. There seems to be a remarkable lack of basic internal flow experiments, to serve as benchmark for validation of codes computing internal flow. The final **Session VII** on intakes and ducts proved, that internal flows are by no means simpler to predict than external flow. But also in this session excellent examples of carefully performed work on code validation were given (6 papers). The additional **Poster Session** was recognized as an efficient means to extend the program of the symposium. 16 papers were displayed, all having been presented in excellent format and stimulating to extensive discussion.

In addition the symposium on computer code validation followed directly a series of preceding AGARD Meetings on directly related subjects :

- o 1986 Aix-en-Provence, France :
  "Applications of computational fluid dynamics in aeronautics".
  (AGARD CP-412)
- o 1987 Naples, Italy:
  "Aerodynamic data accuracy and quality : requirements and capabilities in wind tunnel testing".
  (AGARD-CP-429)

The Fluid Dynamics Panel had realized the increasing role of CFD in aeronautical engineering already in 1986 by forming a standing subcommittee on " Computational Fluid Dynamics" which plays now a supplementary role to the traditional "Windtunnel" subcommittee.

In the following sections, comments on the papers presented in each session are given, followed by a short summary. At the end of this Technical Evaluation Report, conclusions are evaluated to what degree the aim of the meeting has been reached, together with recommendations for future activities in the related field.

3. SYNOPSIS OF THE PAPERS

3.1 Session I - CFD Validation Concepts

The three invited survey-papers of the first session were prescribing the objectives of the symposium and they gave a precise definition of what the program committee was expecting as an outcome of the meeting. The first defined a general philosophy for "Validation" and the remaining two papers were concerned with the experimental and numerical accuracy requirements for validating CFD codes.

PAPER 1. BRADLEY, from General Dynamics, Co-Chairman of the program committee, described a five-phase development cycle for CFD, to achieve a "mature CFD capability", which is understood as a code, that can be used "routinely in complex design applications by engineers." He followed the outcome of a recently performed NRC study on "Current Capabilities and Future Directions in Computatiuonal Fluid Dynamics." A clear distinction was made between "Validation" and "Calibration" which was not always so clear in the past. Four categories of experiments are required to achieve a mature CFD Capability : Experiments to **understand** flow physics (1), to **develop** physical models (2), to **calibrate** CFD codes (3) and to **validate** CFD codes (4).Bounds of errors, for numerical CFD solutions as well as for experimental data have to be evaluated carefully.

PAPER 2. MARVIN, NASA Ames, defined the role of the experiment in the development of CFD for aerodynamic flow prediction. Strong coordination between computational and experimental work is required. It is no longer sufficient to use experimental data from surface or integral measurements alone, to provide code validation and he stated : flow field and boundary condition measurements are emerging as critical additional tasks. He concluded with some examples of "benchmark experiments" performed recently at Nasa Ames Research Center : turnaround duct experiment (1), transonic wing and wing/body experiment (2), 3D supersonic shock interaction experiment (3) and hypersonic all-body experiment (4). There is a major challenge to establish appropriate high Reynolds number, high Mach number and high enthalpy experimental facilities.

PAPER 3. BOERSTOEL, NLR Amsterdam, dealed with the numerical accuracy assessment required for CFD code validation. He divided his contribution into four major parts : specification of primary requirements for numerical flow field simulation concerning flow physics, configurations of interest, initial and boundary requirements (1), development of validation techniques applying logical and numerical verification of the process mapped in the computer codes, verification of accuracy requirements by carefully designed test cases (2), survey of major numerical issues as grid generation, differencing alternatives, numerical viscosity, shock discontinuities, time integration to achieve **consistency, stability and convergence** of a numerical solution (3) finally he reported on the results of recent validation exercises (4), (Agard WG07, AIAA Workshop, international Vortex Flow Experiment).

General Comments on Session I

A very well defined outline of the scope of the symposium was given by the first three papers. As the following sessions show clearly, they have defined a rather high-level standard of the topic "Validation". **First** the validation has to be done numerically by systematic "numerical experiments" but there are pricipal limitations, inevitable by (1) the choice of equations to prescribe the flow physics, (2) the application of a numerical algorithm, (3) the discretization of the flow field (4) and last not least by the technique of modeling viscous effects. **Second** the comparison with experimental data should follow, but be careful : there are equations for which no physical flow exists, (e.g. Potential-, Euler equation) therefore main emphasis should be devoted to differences between CFD and Experiment rather than to agreement. But even for viscous flow codes, experiments are not always available (Reynolds-,Machnumber). So flight tests should have been mentioned, otherwise the validation process would be done for unrealistic conditions. As all participants have noted, the driving motivation for code validation comes from industry. Also a strong requirement was missed : the definition and investigation of sensitivity parameters introduced in a certain "numerical flow" code, to simulate critical, dominating features of physical flow like vorticity, viscosity etc.

3.2 Session II - External Flow I : Airfoils

Five papers were presented in Session II, four of them were reporting on Navier-Stokes solutions, dealing with the influence of discretization, numerical algorithm and turbulence modeling. Two papers were concerned also with full potential flow codes, coupled with boundary-layer methods to take account of viscous effects. Experimental results were compared extensively with computational data. No written handout was available for paper #8.

PAPER 4. FULKNER, WEEKS and ASHILL from the RAE in Bedford reported on a database, obtained by a series of wind tunnel experiments on two (unswept and swept) airfoil models, with systematic exchangable modifications of the rear part of the section from 65% chord to the trailing edges, to allow profiles of different rear camber and base thickness to be tested. Transition had been controlled and comparisons were presented between experimental data and numerical results obtained by three versions of the Viscous Garabedian Korn (VGB) method. The higher order version of the code-family, BVGK, which includes curvature effects and Reynolds normal stresses, shows for the unswept model nearly perfect agreement with measured pressures even for transonic speed with embedded shocks. Also the integrated values for the drag coefficient differed less than 3% from experiment! This was demonstrated at a considerable lift $c_L = .6$ and beginning separation from the trailing edge region. Also for the swept test case the same method (with simple sweep correction) showed excellent agreement with measured pressures, better than the especially for swept wings developed code version SWVGK, which takes account for the crossflow effects. Remarkable close agreement was also shown for the aerodynamic coefficients including drag versus Mach number.

PAPER 5. RUMSEY and ANDERSON, NASA Langley, presented results of an extensive parametric study on steady and unsteady turbulent flow using an upwind differencing, implicit scheme for the numerical solution of the thin-layer approximation of the compressible Navier-Stokes equations. The systematic variation of parameters applied to the algorithm included grid and time-step effects (1), flux-vector and flux-difference splitting techniques (2) and equilibrium and nonequilibrium turbulent boundary layer models (3) for airfoils with and without flow separation. Although the Baldwin-Lomax and the Johnson-King turbulence models were developed originally for the computation of steady flows, both are employed also for unsteady flows sucessfully. The agreement with experimental data lies for the steady cases within 1% for c_L and 3% for c_D and requires a mesh resolution of a 265x101 grid with an extension to the outer boundary to 15c. For the unsteady cases, a 129x49 grid was sufficient to obtain good agreement at Re=4.8 million and Ma=.6. For all cases with separation, the Johnson-King turbulence model yields results in better agreement with the experiment than the Balwin-Lomax model.

PAPER 6. VIEGAS, RUBESIN from NASA Ames and McCORMACK from Stanford University discussed results of an early stage of a program to "calibrate and validate a computer code being developed to compute the performance of circulation controlled airfoils." The code to solve the compresible, Reynolds-averaged Navier-Stokes equations was an implicit, finite volume method, using flux splitting technique and two turbulence models (Baldwin-Lomax and Jones-Launder) have been introduced. The "reference test-case" was a double symmetric airfoil with a jet flowing tangentially over the blunt trailing edge. This rather simple geometric shape turned out to have a complex flow structure because of free shear layers where the jet and the boundary layer merges. In addition, boundary layer separation occurs at the aft end of the airfoil due to the high streamwise curvature and the adverse pressure gradient. The relatively high curvature of the surface required also modifications of the turbulence models. A zonal approach was used, matching the outer region of the flow field (using experimental or numerical data) with the Navier-Stokes computational domain at the rear part of the airfoil. Although the windtunnel walls were represented in the numerical code, the computational results differ significantly from experiment for both cases of different jet momentum coefficients.

PAPER 7. STOCK,HAASE and ECHTLE, Dornier Friedrichshafen, presented results from two computational methods used at industry for viscous flow around airfoils at transonic speed. For the RAE 2822 section, Test Case No. 9 of the AIAA "Viscous Transonic Airfoil Workshop", 1987, the influence of different turbulence models (Baldwin-Lomax, Cebeci-Smith) and the impact of meshgrid refinement and meshgrid adaption was investigated, using a finite volume N.S. code. The results obtained compare quite well with experimental pressure distributions and overall values for lift and (with some overprediction) drag. 200 points on the airfoil surface have been used to compute results from a FPE+inverse Boundary-Layer procedure on the airfoil section DoAL3. The results show also good agreement with the N.S. results, obtained with considerable higher computing cost and with experiments in the range of flow without extensive separation.

PAPER 8. GRASSO from the University of Rom reported on recent work at Rom, Italy, to use local embedded adaptive mesh-refinement of coarse grids for the solution of N.S. equations using a k-ε Turbulence model including compressibility effects. He showed remarkable good results obtained with relatively small numbers of meshgrid points (100 x 20). No written handout was available.

General Comments on Session II

An extensive experimental data-base exists for airfoils. Initial test conditions and geometric boundaries are prescribed and known for the numerical simulation. In some cases the windtunnel wall boundary-layer flow quantities are measured, so they can be represented exactly in numerical procedures. In terms of the previous defined four classes of experiments (Session I), tests for calibration and validation of CFD codes have been made available to the theoretician. Paper #4 and #5 gave a good assessment of the kind of work the program committee had expected to see. Not surprising to an aeronautical engineer working in industry is the message, that boundary-layer methods interactively coupled with inviscid flow prediction are still successful and economically in use. They produce results in good agreement with the present state of N.S. computations at much lower cost. There is a big problem with the prediction of the transition location and therefore all comparisons with experiment are performed for tests with "fixed transition". This problem plays the same important role for boundary-layer codes as well as for the N.S. solvers. So the prediction of transition as a function of Reynolds-number is the weak point in viscid/inviscid interaction prodedures (as for N.S.codes). This is even true for the 2D application on "simple" airfoils, leaving the question open, whether 2D viscous steady flow exists in nature at all. The presentations in Session II have all circumvented the problem of predicting transition.

A great potential seems to exist in local meshgrid refinement techniques. The total number of meshgrid points gives no guarantee for a good numerical solution, it says nothing - local high-density of flow field discretization in critical flow regimes (surface curvature, shocks, shear layers etc.) is necessary to achieve economic flow field prediction at sufficient accuracy and at affordable cost.

3.3 Session III - External Flow II : Vortex Flows

A simple geometric shape again - but a rather complex flow field, that was the subject of this session on delta-wings and delta-wing-bodies. These flows are characterized by vortical type flow separation at the swept leading edge starting at moderate angles of attack. Assuming that vorticity and not viscosity dominates the structure of this type of separated 3D flows on delta wings, the highest level of equations prescribing inviscid flow has been attacked. Flow codes solving the Euler equations became available during the past decade and they have been applied rapidly in industry to achieve nonlinear design work.

PAPER 9. ELSENAAR, NLR Amsterdam, HJELMBERG, FFA, Sweden, BÜTEFISCH, DFVLR Göttingen and BANNINK from the Delft University gave an (invited) survey on a recently performed international effort, to establish an experimental database for a simple 65° delta-wing-body windtunnel model, referred to as "International Vortex Flow Experiment". In addition to pressure and force measurements, surface flow was visualized, and a complete flow field assessment using laser anemometry was provided. Also configurational aspects (e.g. round versus sharp leading edge, canard interference, unsymmetric flow conditions) have been investigated. Special attention has been given to experimental details effecting the performance of numerical flow codes: Boundary layer transition and Reynolds-number (1), numerical resolution in the L.E. region (2), body influence (3) and wind tunnel wall and support interference (4), vortex stability and burst (5) and vortex shock interaction (6). Two models of different scale have been built and tested in three wind tunnels in the Machnumber range from .4 - 4.0. in addition, several smaller research models have been manufacuted and tested in research institutes and universities. So the presentation included comparison of experimental data obtained in different windtunnels for identical and for different models measured at overlapping Reynoldnumbers. The agreement for the same test conditions is good for the total forces (except c_L-max.) and for the model with sharp leading-edge, but differencies exist for round leading-edge in the comparison of measured pressures.

PAPER 10. WAGNER, HITZEL from Dornier, SCHMATZ, SCHWARZ from MBB and HILGENSTOCK and SCHERR from the DFVLR reviewed in the following paper the suplementary contribution of CDF in Germany to the above mentioned Vortex Flow Experiment. Three different Euler codes were used extensively to simulate the flow numerically but also first results of N.S. flow solvers were presented. All work sofar is based on the Stockholm FFA IVFE meeting in fall 1986. Main emphasis was devoted towards the influence of L.E. geometry on vortex formation (1), formation of shock waves in transonic vortical flow (2), flow structure of vortex break down (3), losses of total pressure in the vortex core (4) and possible improvements by solving the time averaged N.S. equations(5). Reasonable agreement was shown with respect to overall forces and (in some cases) to pressure distribution in the case of sharp L.E. Although the mechanism of the high pressure losses in the vortex core and of the vortex breakdown is not completely understood, some similarity between calculation and experiment has been demonstrated. But obviously, sufficient mesh resolution has not been achieved in some applications. This is especially true for the cases with the round L.E. and the first results obtained by N.S. Solutions. So more detailed analysis is required, and the comparisons between measured and computed flow fields are still missing.

PAPER 11. NG and NELSON from the University of Notre Dame, US, investigated the phenomenon of vortex breakdown over slender delta-wings with parametric variation of L.E. sweep between 70° and 85° experimentally, using laser anemometry and seven-hole pressure probes. Specifically data were presented on vortex trajectories (1), wake surveys (2), swirl angles before and after vortex breakdown (3) and effects of Reynolds-number on vortex breakdown(4). Most interesting is the observation made during this windtunnel tests, that both forms of vortexbreakdown exists on the same model (spiral and bubble type). A correlation of the measured swirl angle with vortex breakdown location does not allow the prediction of vortex burst from swirl angle alone.

PAPER 12. FULKNER and ASHILL from the RAE, UK, reported on extensive comparisons between measured and calculated data on a more realistic wing-body geometry representing a (simplified) typical configuration of present day combat aircraft for supersonic flight. Theoretical results were obtained by a finite volume technique, solving the time dependent Euler equations and applying multiblock structured grids. Three different combinations of grid topology ("C"-, "O"-, "H"-type) applied for wing and body have been ("coarse" and "fine" grid) investigated systematically and not all discrepancies observed are fully understood, (e.g. the fact that the type of grid used at the forebody(!) changes the drag polar significantly). Nevertheless the comparison of measured and calculated overall coefficients for lift and drag show a remarkable agreement even up to high c_L/c_D, for two different wing designs. Special attention is given to the capability of CFD to predict changes in pressure distribution due to small changes in geometry and to integrate increments for lift and drag. This effort shows only trends which look similar. Also the prediction of pitching moment from CFD in reasonable agreement with experiment remains still an unsolved problem, but also in this case, the comparison of components, like zero-lift pitching moment of two different wing designs looks quite promising.

General Comments on Session III

As previously mentioned, a simple shape by no means guarantees a simple flow structure. Experimental data bases have been established in Europe and US for delta wings and delta wing bodies providing in addition to forces and pressures also detailed measurements of the surrounding flow field for the highly nonlinear region with L.E. vortex flow separation. It has been demonstrated that the highest level of inviscid flow codes can provide numerically results which are in many cases in reasonable agreement with experimental data, although many details of the flow like vortex breakdown or vortex/boundary layer- and vortex/shock- interaction are still not properly modeled in CFD codes. Also the influence of grid resolution requires further work, ("local" grid density), in order to bring first results of different flow codes in agreement. But there are other critical remarks raised from the discussions. The **validity of comparisons** of results of an inviscid code like the Euler solutions with experiments is at least questionable - no Euler-type flow exists in reality, so there are some assumptions : Vortex flow is dominating(1), there is no vortex-boundary-layer interaction(2) and no shock-boundary-layer interaction(3). So the question is not why discrepancies in comparisons of theory with experiment exist - the question is why there is agreement! As first results have shown, N.S. solutions surprisingly do not yet improve at present the situation.

3.4 Session IV - External Flow III : Wings/Wing Body

The area covered by 7 papers contributing to this session is widespread and reaches from detailed flow measurements within the boundary layer and boundary layer code validation (#17) to a compilation of results obtained by CFD codes solving various forms of the potential equations (#14,#15,#18) to Euler and N.S. solvers (13,15). In addition two papers have been reporting on industrial aspects of CFD code application during "real" project work, without reporting on the validation process of the codes having been used (#16,#19). As in the previous sessions new data bases for code validation have been established, but not extensively used by numerical analysis.

PAPER 13. KORDULLA, SCHWAMBORN and SOBIECZKI from the DFVLR in Göttingen gave a detailed description of the so-called DFVLR-F5 wing experiment being considered to be a novel step towards the rigorous validation of N.S. codes. This experiment is especially designed for code validation by the determination of flow conditions on the entire surface of the prescribed control volume, including wind-tunnel walls and measurement of the flow field at the entrance and exit plane. Although the configuration of the wing is considered to be rather simple (Aspect ratio 9, symmetric cross sections, small angle of attack 0° and 2°) the numerical simulation turns out to be a hard exercise for the computational aerodynamicist. A workshop has been organized in 1987 and computed results have been presented for 6 selected testcases by 6 different participants (NASA, FFA, NAL, MBB, DFVLR). The compilation of numerical data show considerable scatter (up to 100% in terms of integrated c_L and c_D), even for the "free-air" test conditions (without wind tunnel wall simulation), in computed pressure distributions at spanwise cross sections. The reasons for this unsatisfactory status were not identified, but there are many sources to be investigated in the near future, like ˈɹbulence modeling, numerical algorithms and grid resolution. The prediction of 3-D transition seems to be the greatest uncertainty in viscous analysis. Another remaining problem is the limitation in memory of present day computers and computer cost which approach now the level of cost for experimental work.

PAPER 14. ANDERSON and BATINA from NASA-Langley showed a completely different approach to flow code validation. They applied Euler and transonic small disturbance (TSD) potential flow equations to compute steady and unsteady flows for 2- and 3-D test cases. The 2-D cases were studied primarily to determine the effects of grid density, spatial accuracy and far field boundary conditions. After intoducing both entropy and vorticity corrections to the TSD method, the same good agreement with experiment was achieved as for Euler calculations of steady and unsteady (even slightly better) 2-D and 3-D flow. Only for the 3-D steady F5-wing test case exist significant differences between measured and computed data (obviously due to shock boundary-layer interaction). A reduction of computer time of one order of magnitude was the driving motivation for "calibrating" the TSD code. In this case the grid remains fixed for steady and unsteady calculations and an extension of the applications to complex configurations seems to be straight forward.

PAPER 15. SCHMATZ, BRENNEIS and EBERLE, MBB Ottobrunn, again presented 2-D and 3-D steady N.S. and unsteady Euler code applications. All codes are based on the unfactored implicit equations in time-dependent form, using finite volume and flux--splitting techniques in highly vectorized programmes. As was expected, the turbulence model represented in the N.S. code, (part I) influenced very strongly the solution if separation occurred. In cases for attached flow the computed results compared quite well with experiment. Unfortunately the code showed only limited agreement with the experiment for the 3-D delta wing (see also Paper #10). Mainly due to the extremely high computer cost, the reasons for this discrepancies haven been left open for future validation work. In part II (which should have been a separate paper), extensive comparisons of computed and measured pressures have been presented for subsonic and transonic unsteady flow about a rectangular and a swept oscillating wing. In addition some results of different codes published in the literature have been included. Main advantage of the solution presented is a better agreement with experiment in the nose region (and in cases with shocks at the shock domain where potential flow fails completely) of the wing sections. To improve remaining differences at the rear part of the wing, more recent investigations are under way to include viscous-inviscid interactions by coupling the Euler solution with a boundary layer code. Most important however for project applications is the question of reducing computer cost. Unfortunately this topic has not been addressed at all.

PAPER 16. HECKMANN from AMD, France, gave a complete survey on industrial requirements for viability and validation of computer codes using wind-tunnels and flight tests. He did not prescribe the validation process of a selected code but he showed results obtained by various programmes available at his company in comparison with experimental data obtained from windtunnel and flight test and he referred to the large amount of data available from flight testing of a number of successful aircraft designs. He underlined the complementary role of theory and test during the aircraft design process and the importance of knowing test boundary conditions for numerical analysis to understand the physics of the flow.

PAPER 17. BARBERIS and CHANETZ from the ONERA, France, returned back to the subject of validation of a specific computer code by carefully performed wind tunnel work. The objective was "to understand better the physics of flow separation phenomena and to provide well documented data for the validation of theoretical models". This paper follows the real sense of the subject of this meeting. Flow field visualization and detailed flow field measurements (including measurements in the boundary layer domain) have been made, using pressure probes and laser anemometry on simple blunt bodies with elliptical nose shapes. In addition, pressures, flow visualization and shear stress at the model surface has been recorded. The inverse boundary layer code used for validation provided also stability for continuation of the calculations in the region of separated flow. Special emphasis was devoted to the phenomena of separation lines at the lee side of the body at incidence and to include a numerical model in the difference scheme solving the boundary layer equations. The results obtained with an algebraic turbulence model are in good agreement with measured data i.e. streamlines, separation lines and boundary layer velocity profiles.

PAPER 18. CARR from ARA, UK, Chairman of GARTEUR Action Group AD(AG05) reported in a similar way like paper #9 and #13 on another computational "Olympic" competition. Test case was the civil transport type DFVLR F4 wing at one subsonic and one transonic flow condition at small angle of attack (design condition, attached flow). Although all participants (10) contributed results from field methods, only surface pressures, local and overall integrated coefficients could be compared with experiment but even this was not included in the presentation. Seven contributions were solving the full potential flow equations, either in non-conservative or in conservative form, applying C-H, O-H and H-H mesh type grids. Finite difference schemes have been included, as well as finite volume and finite element techniques.

Further differences exist in the exention of the computed domain and the definition of the outer boundary condition. The remaining 3 contributions came from Euler solutions, all finite volume schemes using C-H Type grids. Reasonable agreement of computational data has been achieved for pressure distributions and integrated c_L (< 1.5%) and pressure drag c_{D_P} (< 14% scatter of computed results). But for transonic flow with a shock on the upper surface of the wing, all 3 classes of solutions show significant scatter, as well for the pressure as for integrated coefficients. Unfortunately, no experimental reference is given for both test cases in the paper! So no validation has been reported, only a compilation of numerical results has been presented. The potential of Euler codes to predict tip effects (this can be the driving argument for using Euler codes for design at attached flow) was indicated but not verified with the (existing) experiment.

PAPER 19. VOOGT, MOL, STOUT and VOLKERS from FOKKER, NL, gave another industrial review on computational methods available in industry, similar to paper #16. They stressed the topics of 2-D and 3-D high lift (transonic small perturbation or panel methods coupled with boundary layer), the computation of intake ducts (inviscid panel methods) and a high speed 3-D full potential flow solver for transonic speed. Several examples were given for each category of CFD codes, mostly in fair agreement with experimental data. However the statement was made, that "empirical rules" must be included in CFD codes to achieve solutions for "practical" flow problems and there is still a need for simple prediction methods for overall characteristics, which have to be used complementary to complex flow codes. Unfortunately these correlation process of including empirical criteria in complex flow codes has not been prescribed in detail.

General Comments on Session IV

The message was, that CFD complex flow codes have reached a rather high standard of application during design and analysis of modern civil and military aircraft. Computations reveal details of the flow, which could not be obtained otherwise. On the other hand, cost for complex flow computation is approching the level of cost of a windtunnel test (at least in industry where cpu time has to be payed at a rate of about $ 1000.-- per hour on a vector computer).

But there are some other summarizing comments on session IV. Experimental data bases including flow field data are as well established for civil type high aspect ratio wings as for delta wing military aircraft (previous session), but in contrast they have not been used extensively sofar for code validation. There is still a great challenge for the theoretician to investigate the treatment of windtunnel walls in CFD (1), to predict transition free viscous flow (2) and laminar and turbulent shock-boundary-layer interaction (3). The conclusion is, that at present even for attached flow, the N.S. results are not validated, perhaps due to the fact, that only a few solutions were available because of the high computer cost. Euler results are generally in good agreement with experimental data, although viscous flow effects (e.g. by coupling with boundary layer codes) have not been included. Further progress can be noticed on the status of various forms of the potential flow equations and we have seen an exellent exercise for validation of such codes by introducing exact relationships for entropy, shock strength and vorticity.

3.5 Session V - External Flow IV : High Speed Flow

The session on high speed flow was relatively short, only five contibutions, but at least three fit perfectly into the subject of the meeting. Three papers were restricted to inviscid flow around slender missiles (Euler solutions). The limited scope of the material shown in the Session stands in remarkable contrast to the huge interest in supersonic and hypersonic projects now under condideration in the whole industrial world for military application (e.g. NASP), civil transport (e.g. HTV, TGV) and space activity (e.g. SÄNGER,HOTOL).

PAPER 20. DEIWERT, STRAWA, SHARMA and PARK, NASA Ames, outlined the scope of the experimental program for validating real gas hypersonic flow codes at NASA Ames Research Center. This program includes tests in ballistic ranges, shock tubes, arc jet facilities and heated air hypersonic wind tunnels. In addition flight test results obtained by the Aeroassist Flight Experiment, the Space Shuttle, Project Fire 2 and planetary probes are referenced.The validation activities for hypersonic flow codes comprises the areas of perfect gas (1), real gas (2), jet plume flow interactions(3), combustion (4), thermochemical (5), non-equilibrium gas (6) and boundary layer (7). Some experiments were prescribed representing the type of work which is still under way but only few numerical data obtained by numerical simulations were compared with the experiment. Eacn of the above mentioned 7 objectives could have been subject of an separate paper! Important to note that not one single experiment is sufficient to validate a given code over a broad range of application. An experimental program using a number of different facilities for specified dominating flow parameters is required.

PAPER 21. DORMIEUX and MAHE AEROSPATIALE/ONERA,France, reported on the development of a exhaust system for direct thrust vector controlled missiles, using a lateral jet blowing out near the the center of gravity. Numerical calculation utilizing a 3D pseudo space marching Euler flow solution have been performed on two selected testcases. First a rectangular jet blowing from a flat plate with M=2.5 in an external Flow with M=2.o and second a slender fuselage (15D) with a circular jet blowing from the lower side at the 8.5D station at 30° incidence with the same Machnumber conditions was shown. The comparison of measured and calculated data show good agreement for Mach-number distributions and the location of detached shocks, but also differences in the position of vortex location and vorticity. Some details like separation upstream and reattachment downstrem of the nozzle was in complete disagreement between experiment and theory due to the neglection of viscous boundary layer effects in the calculations. Nevertheless the computational results compared sufficiently well with experiments to predict normal force coefficient for two missile configurations have been sucessful.

PAPER 22. GUILLEN and LORDON from AEROSPATIALE, France, gave a complementary paper to the previous one. They described the validation process of two versions of an Euler flow code (space marching and time marching) for applications to missiles at incidence and separated supersonic flow. In contrast to the rather high standard achieved for swept wings (Session III), at least for wing configurations with sharp leading-edges, the situation for the prediction of vortex separation with Euler codes for smooth slender bodies is worse. Inviscid flow codes are not able to model correctly flow separation on a curved surface. The data of separation line have to be obtained empirically and have to be introduced in the numerical procedure like using a Kutta condition. In addition a shock fitting mesh grid technique was applied and, to stabilize nolinear effects due to shocks, "numerical viscosity" has been introduced in the code. All these parameters of major influence have been investigated carefully, so real code validation is reported by the present paper. The conclusion was, that two different kinds of seperated flows can be distinguished : at high cross flow Mach numbers, the computed results seem to be independent of most parameters investigated (numerical method, numerical viscosity, position of separation line). For lower supersonic Machnumbers, the computed results depend strongly on the parameters mentioned above.

PAPER 23. SHEPHERD and TOD from BAe, Filton, UK, gave a paper on a multiblock Euler code procedure for application during the development of weapons in the project environment. The code developed closes the gap between existing subsonic panel methods and the space marching Euler codes which work for the high Machnumber region. Validation of the flow code was presented for three simple bodies. Good agreement with experimental data was reported for cases of small incidence, so no leeside vortex separation has been treated. First results for a more complex body--fin missile configuration compare quite well with measured data for the overall coefficients c_N (< 7.7%) and c_M (<12.7%) at 8° AOA and .7 < M < 1.45. But a lot of future work following the aspects treated in Paper #22 is still needed to provide a satifactorily working design tool for the aerodynamic engineer.

PAPER 24. ZWAN, CROOKS and WHATLEY from GD, US, finally gave an example for validation of surface catalycity using a numerical viscous shock layer approach. Unfortunately no written handout was available, so the evaluator has to rely on his short notes. But the paper represented this type of work, the program committee had in mind. Validation of the numerical procedure has been reported in four steps. First the code development was described, second a carefully selected testcase was investigated, third the comparison of experimental and theoretical results has been shown and fourth, conclusions have been drawn out from this comparison, to improve the models applied in the numerical procedure to simulate the physics of the flow.

General Comments on Session V

We had two sessions in the FDP HYPERSONICS Symposium in spring 1987 in Bristol, UK, concerned with computational aerodynamics for high speed flows and code validation. So the present session could be considered as a complement to the Bristol meeting, where real gas, non-equilibrium viscous effects and heat transfer were primarily treated. The evaluator of the Bristol sessions complained most about the lack of carefully designed experimental data bases for validation of existing CFD codes. This situation has not been improved to much by session V in Lisbon. There is an additional special need for detailed experimental studies concerning flowfield and separation line measurements. Interesting to note, that no validation of tubulence models for the use in N.S. codes has been reported for high speed flow. Where is the "International Supersonic Flow Model" designed for CFD ? Nevertheless the evaluator feels that in spite of the missing validation, a couple of useful engineering applications of CFD have been demonstrated. This concerns vortical flow interaction with lifting surfaces, prediction of aerodynamic coefficients (with exeption of drag) and analysing jet-surface interference effects.

3.6 Session VI - Internal Flow I : Turbomachinery

This session was opened by an excellent introduction to the meaning of validation of computer codes and there was at least one more paper on validation in that true sense. At least four papers were not aligned properly to the subject of the meeting, being more or less restricted to the report on applications of existing CFD codes. Internal flow simulation by numerical techniques turns out to be by no means easier to solve than external flow in spite of better defined outer boundary conditions and limited computational domain.

PAPER 25. POVINELLI from NASA Lewis described validation experiments designed for research in three areas : inlets, ducts and nozzles (1), turbomachinery (2) and chemically reacting flows (3). Strong criteria were applied both to experimental and numerical work : "Validation may only occur when the accuracy and limitations of both the experimental data and the code's algorithm are known and understood over a range of specified parameters" and "it is critical that the users be aware of the code validity and limitations". Critical physics for inlets, ducts and nozzles to be modelled were assigned to three major classes : highly 3D flow fields (1), shock boundary layer interactions (2) and shear layer control(3). Experiments investigating (2) and (3) were presented in the paper and also three major categories of CFD codes (3D subsonic PNS, 3D hypersonic PNS and full 3D NS) were validated using the experimental results. Consequently test set-ups and numerical approaches were prescribed for the other two groups of technical applications, turbomachinery and chemically reacting flows. Numerical modeling and the experiments in this paper were exactly of the type required for the CFD assessment and code validation process, the declared aim of the meeting.

PAPER 26. KOSCHEL, VORNBERGER from the University of Aachen and LÖTZERICH from Dornier reported on the development of an explicit finite element scheme (FEM) solving the Euler- and Navier-Stokes equations. A special adaptive grid refinement technique allowed local refinement of the grid. "Validation" was achieved by comparisons of computed results with experimental data obtained on a cylinder and a sphere, shock reflection at a wall, oblique shock wave interaction with a laminar boundary layer and the internal flow through an inlet and a turbine cascade. Sofar "satisfying agreement" of computed and measured data was reported for the 2D applications. In this paper the main emphasis was given to the investigation of the effects of local grid refinement and convergence to steady state solution of the FEM approach, which has been used in place of the more familiar finite volume scheme. An attractive feature of the code allowed the investigation of viscous effects by switching from "Euler" to Navier-Stokes". This has been done for several test geometries. 3D Validation seems to be in progress.

PAPER 27. THIBAUD, DROTZ, and SOTTAS from the Swiss Federal Institute of Technology presented a paper on the numerical flow simulation for hydraulic machines (Francis turbine) based on the solution of the incompressible steady Euler equations. Coriolis, centrifugal and gravitational forces and vortical type flow were included but no viscous correction has been attempted. The influence of systematic mesh grid refinement was demonstated and "Good qualitative Agreement" with experimental data was reported, but the quality of measured data was not clear (5 hole probes) and boundary layer effects should cause some local differences. Nevertheless the procedure has already been introduced in the design and optimization process of new turbomachines.

PAPER 28. LAKSHMINARAYANA, KIRTLEY and WARFIELD from the Pensylvania State University gave a rather extensive review on a system of CFD codes developed to solve the incompressible Navier-Stokes equations for internal turbulent flows. Five computational approaches have been chosen : singlepass space marching (1), space marching with a Poisson pressure solver (2), multipass space marching (3), time marching (4) and zonal technique (5) combining parabolized N.S. with an Euler computational domain. Main emphasis has been devoted to calibrate computed results with a series of existing exact analytical or very accurate experimental data (for laminar and turbulent flow) on simple test geometries. Simple configurations are considered to be essential to detect errors associated with grid and boundary conditions and, to start with the numerics, also laminar flow is essential to avoid the problems with tubulence modeling. Subsequently six different configurations (90°-duct,S-duct,cascades and rotors), specifically designed to provide very accurate data for code validation, have been selected to validate computational results obtained by the above mentioned codes. Because of limitations in cost and time it was not possible to run all codes for all configurations, so only some results of representative cases were presented. The impression is, that this type of work has to be continued. Especially the influence of turbulence modeling (Baldwin-Lomax versus k-ε Model and others?) need further validation. It is felt by the authors and the evaluator, that there is a lack of benchmark-quality data on complex internal flows like turbine end wall and rotor flows, but also new measurement techniques are required for resolving e.g. wall layers and corner flows. These measurements are not only complex but also very expensive.

PAPER 29. MARESCA, FAVIER, NSI MBA and BARBI from CNRS, Marseille, reported on comparisons of experimental and computational data obtained on helicopter-rotors (hovering and forward flight) and propellers (forward flight). Computer codes were based on potential flow lifting line theory without vicous flow corrections. The comparison of the distribution of circulation and velocities in spanwise direction across the rotor blades shows significant scatter. Only by prescribing circulation a priori, better agreement could be achieved. In some cases the overall thrust coefficient could be predicted well. Validation of the computer codes has not been achieved in detail, nevertheless engineering applications are promising.

PAPER 30. ERIKSSON and BILLDAL from the Norwegian Institute of Technology gave a rather detailed report on the development of a time-marching numerical (cell centered finite volume) scheme applied to solve the Euler/Navier-Stokes equations for 3D turbomachinery flow. To validate their results, very detailed flow measurement by Eckardt (DFVLR,Cologne) on a centrifugal compressor impeller at high speed were chosen. Velocity vectors and velocity contour plots are presented in detail for experiment (1), inviscid Euler (2), viscous thin-layer Navier-Stokes (3), and viscous full Navier-Stokes (4) simulation. The analysis of data presented shows the limits of applicability of inviscid flow solvers. The results of the viscous flow model are closer to reality in spite of the use of a "primitive turbulence model". But more important, at least for the test case presented, the thin-layer approximation gives nearly the same level of agreement with the experiment than the full Navier-Stokes solution. In this respect, figures on the computer time needed to obtain the data in each code-class could have been informative.

PAPER 31. HAPPEL and STUBERT from MTU, Munich, showed results from the computation of inviscid transonic 3D turbomachinery (single stator or rotor row) flow using a time-marching Euler code. Pressure distributions and Mach number contours on the blade surface and in the region from blade to blade are shown in comparison with experimental data for the stator but no measured values were available for the rotor selected. No more validation has been attempted yet.

General Comments on Session VI

As could have been expected, the session was rather inhomogenous. Putting turbines, compressors, hydraulic machines, propellers and rotors in one session, the outcome must be confusing.
But some conclusions can be drawn in common :
(1) As previously mentioned, internal flows are by no means easier to handle for CFD than external flows. This was not so clear from the beginning, because some of the boundary conditions (at the wall) are better defined in real flow and therefore easier to represent in the computer code. Also the computational domain is limited in the outer extension, so in principle, memory requirement could be less extensive.
(2) For the computation of internal flows additional inputs concerning the onset (inflow) flow conditions have to be specified precisely. This requirement is not always easy to fulfill.
(3) Specification of outflow in combination with some inflow conditions is sometimes necessary to control massflow.
(4) The use of inviscid numerical flow simulation (e.g.Euler) for internal flow problems is at least questionable. In most cases, channel flow with large viscous shear layer regions and viscous wakes dominate the physics of the flow.
(5) Because of the lack of experimental data bases (expensive to achieve) calibration and validation of CFD codes have been done only to a very limited extent.
(6) Nevertheless there is a strong demand from industry to provide computer codes for analysis and design in turbomachinery and for engine-airframe integration.

3.7 Session VII - Internal Flow II : Intakes and Ducts

The session concluded the symposium but should have been arranged before session VI. With one exeption the papers presented were dealing with Navier-Stokes solutions, mostly comparing computed with measured data. Paper #34 was a good reference for the complementary role of theory and experiment in turbulence modeling. But two papers were not concerned with internal flow at all.

PAPER 32. BUERS, LEICHER, from DORNIER and MACKRODT from DFVLR, Germany, gave an overview on the typical industrial application of an Euler code with simulation of boundary layer effects to a supersonic side mounted inlet for a fighter aircraft forebody geometry. A wind tunnel model has been carefully designed and a ramp bleed system has been installed to stabilize the inlet flow. Using Euler computations, the boundary layer thickness at the ramp was calculated. This determined the quantity of bleed installation, necessary for the wind tunnel test specification. Only a few comparisons of computed and measured data have been shown in the paper. By simulation of boundary layer effects using an "equivalent source concept" at the inlet ramp, the agreement of pressure distributions at the ramp with experiment could be improved significantly. But this was not code validation in the "true sense" of this symposium. Intake flow fields should have been measured during wind tunnel tests but at least in the paper presented, there was no attempt to validate the Euler code with the test data.

PAPER 33. SEMMES, ARBITER and DYER from Wright Patterson AFB, US, reported on a 2D axisymmetric Reynolds averaged N.S.Code which was selected to analyse internal flow problems. Results were compared with experimental data, to "determine the extent of usefulness". The test configurations were : 2D hypersonic inlet (1), axisymmetric condi-nozzle (2) and an axisymmetric subsonic diffusor. The code was able to predict good agreement for surface pressure distribution for all Mach numbers, but in some cases it fails to predict boundary layer profiles correctly, this was specially true in regions near flow separation. This can be attributed certainly to the turbulence model used (Baldwin-Lomax), sofar the authors recommend further calibration tests.

PAPER 34. KIND, YOWAKIM and REDDY from Carleton University, Ottawa, prescribed a test set up for the investigation of swirling flows in a cylindrical annulus. Measured data include static and dynamic pressure distributions, axial and tangential wall shear stress components and profiles of the mean velocity components of the Reynolds shear and normal stresses. Consistency checks indicate, that the data are of high quality and sofar suitable for validation of computational codes. A simple mixing length turbulence model was used to compute the development of turbulent flows. Before introducing the turbulence models in the code, the numerical procedure was checked carefully for laminar flow where exact solutions were available.

PAPER 35. BARATA, DURAO, HEITOR from Instituto Superior Técnico, Lisbon and MCGUIRK from the Imperial College in London reported on a special test arrangement in a water tunnel. An axisymmetric jet was directed to a wall after penetrating a cross flow stream. The velocity field (u and v mean and turbulent velocity components) was measured by a dual-beam forward-scatter laser velocimeter and the turbulent velocity components were also used to compute local shear stress distribution. Experimental results have been compared with numerical data obtained by two 3D full Navier-Stokes codes using the k-ε model. The compilation of data shows general agreement for the evaluated mean velocities, but deviations for the shear stress inside the impingement zone, which shows the limitation of the turbulent viscosity hypothesis introduced by the k-ε approach.

PAPER 36. CIOFALO, University of Palermo and COLLINS from the City University of London gave a paper on the numerical simulation of the starting flow of an incompressible fluid past a downstream-backfacing step. The investigation was of primarily numerical nature, concentrating on effects of boundary and initial conditions, differencing scheme, time stepping and other "computational parameters" on the solution of the 2D Navier-Stokes equations. A compilation of a couple of experimental data from published literature is used to compare with the results (mainly reattachment length and transition to turbulence after separation) obtained by the new code. Special emphasis in the present paper was devoted to the starting and developing flow and only one experiment was found to be available for comparisons with theoretical results. After extensive numerical tests, stability in terms of grid size, time step and convergence was achieved. Comparisons with experimental data were restricted to reattachment length, sofar validation was restricted to observe some similarity with flow visualisation.

PAPER 37. RUDI, KUMAR, THOMAS, GNOFFO from NASA Langley and CHAKRAVARHTY from Rockwell, US, continued, dealing with four computer codes for solving the compressible Navier-Stokes Equations. To evaluate the range of validity of the solution procedures and the physical models in each code the paper recommends 3 steps : internal consistency checks (1), comparison with other existent codes (2) and comparison with experimental data. Using existing CFD codes, the first step was assumed already having been completed by the individual programmer. Therefore three different test problems, typical for high speed internal flow, were selected for application of the codes, for comparing the data with each other but also to some extent to experimental results : supersonic flow between two walls (1), a hypersonic inlet (2) and a 3D corner with supersonic flow (representative to a scramjet inlet geometry). Even for the simplest case, the 2D supersonic laminar flow between two walls, the computed results for skin friction differ significantly, but unfortunately no experimental data were available for "Validation". Mach number profiles for the transitional and turbulent flow at the hypersonic inlet cowl were surprisingly well predicted in comparison with the experiment. This was also found for the wall pressures on the centerbody and the inlet cowl, but differences occur for the pitot pressure profiles in cross-sections of the intake channel flow. Only few data were available for the 3D supersonic turbulent corner flow. So further validation is certainly needed.

<u>General Comments on Session VII</u>

Most of the remarks to the preceeding session VI hold also for this last session of this symposium, which was concerned with "Validation" of computer codes. So they should not be repeated again. Specially the feeling of the evaluator that internal flow is not simpler to compute than external flow was amplified. In any case the "external flow people" seem to be better organized than the "internal flow engineers". So the definition of Benchmark-Test-Cases was missing and there is obviously no Data-Base available for internal flow calculations. Also the organisation of international workshops and action groups was not reported during this meeting. Two major activities of AGARD remained without reference: the ongoing Working Group 13 of the FDP "Air Intakes for High Speed Vehicles" (advisory report scheduled for 1990) and the PEP WG 18 on "Test Cases for Computation of Internal Flows in Aero Engine Components". The latter is on delay, an advisory report can not be expected to be published until the end of 1989. As an outcome of the somewhat unbalanced activities concerning external/internal flow the AGARD FDP should perhaps consider to continue with symposia dealing with the subject of CFD (1986 "Applications", 1987 "Hypersonics" and 1988 "Validation"), e.g. 1991 "CFD for internal flow problems"?

4. POSTER SESSION

In response to the Call for Papers for the present Symposium more than ninety abstacts have been submitted to the program committee. Only 39 could have been included to the official 3 and 1/2-days program. One way to avoid the need of rejecting too much proposals was the organisation of a "Poster-Session" within the schedule of the meeting. The 19 papers which were finally accepted and 16 of them presented in this session were not considered to be "second-class" papers and the decision was made to include them as "full-length" papers in the final proceedings.

POSTER PAPER 1. AMENDOLA, TOGNACCINI from Aeritalia and BOERSTOEL from the NLR introduced a computer-program system for the numerical simulation of subsonic and transonic flows around complex (propeller-wing-nacelle) configurations, solving the Euler equations on multi-block grids. Modularity has been applied to the program, the main parts were a block decomposer (1), a grid generator (2), the flow solver (3) and postprocessing routines (4). Propellers have been modelled as an actuator disk specifying discontinuities. The accuracy of the computational results are further to be analysed, experimental data were found in good relation with the numerical results.

POSTER PAPER 2. BANNINK, HOUTMAN and OTTOCHIAN from the University in Delft reported on turbulent boundary layer calculations on a 65° delta wing at transonic and a circular cone at supersonic speed. For the program, conical external flow is assumed and the solution marches in cross-direction from the reattachment line. Experimental pressure distributions (which are not always available) are used to generate the inviscid solutions at the edge of the boundary layer. The aim was to compute the surface flow and to predict the separation lines on conical bodies. But for real application some essential restrictions have to be made : no embedded shocks, no upstream influence of trailing edge and wing tips, no vortex breakdown and exclusion of the transitional region.

POSTER PAPER 3. FAVINI, SABETTA and ZANNETTI from the University in Rome showed results from the design and the validation of a 2D Euler code. No written handout was available to the evaluator.

POSTER PAPER 4. PEREIRA from Instituto Superior Técnico in Lisbon and DURST from the University of Erlangen-Nürnberg presented results from a numerical solution of the 2D Navier-Stokes equations. The code has been validated by comparisons with "carefully selected laminar steady flows" (backward facing step and flow over a fence at different Reynolds-numbers) where laser-doppler experimental data were available. The influence of different numerical discretization schemes for convective terms was extensively investigated. For unsteady recirculatiing flow calculations on a sudden pipe expansion and a square cylinder, good agreement of numerical and experimental data was reported, all restricted to laminar flow.

POSTER PAPER 5. FIRMIN and MCDONALD from RAE gave a survey on activities within the GARTEUR Action Group AD(AG07). A low aspect-ratio wing has been designed and test will be made in the low speed facilities of the NLR (LST, 3.0x2.25m) and ONERA (1.8x1.4m) using separate physical half-models to explore the 3D shear layers on the upper and lower surface of the model. The design conditions were quite challenging because of the requirement for (1) extreme three-dimensionality within the boundary layer and (2) incipient separation near the trailing edge on the upper surface. This is a somewhat unique feature of this specific project, because nobody has undertaken up till now the design of a wing for separated flow. In order to examine the successful design for incipient separation, a third (Pilot-) model has been built and tested in a small low-speed tunnel at the NLR. A compilation of calculated data was presented, obtained by CFD codes available at different locations (BAe, ARA, NLR, ONERA), showing quite an amount of scatter for the shape factor and the twist within the boundary layer. Out from this GARTEUR activity an extremely valuable data-base for future code validation could be expected. Paper relevant to Session IV.

POSTER PAPER 6. BALTAR and TJONNELAND from Boeing presented results from extensive comparisons of computed and measured data obtained on slender cones in hypersonic flow (M=18.7). Theoretical base was a zonal solution approach using full Navier-Stokes and Parabolized Navier-Stokes modules.Plots were shown comparing predicted and measured shock locations, surface pressures, surface heat transfer and other flow field properties. The agreement between theory and experiment was considered to be "fair" taking in account that some physical properties of the flow were not represented accurately in the test section. Paper relevant to Session V.

POSTER PAPER 7. FORTIER from the DRE, Quebec, reported on the development of a panel-method using vortex singularities to solve the linearized potential flow equation for sub- and supersonic flows. "Validation" was attempted by comparing computed results (only integrated coefficients) with experiments for simple (ONERA M6, AGARD Model B) and more complex shapes (USAF guided weapon, NASA cruise missile, CF-18A Aircraft). No viscous flow correction was undertaken. The relation between computed (inviscid) and measured data was adequate. More sophisticated (higher order) panel codes with viscous flow correction (or even flow separation) are state-of-the-art in industry. Paper relevant to Session IV.

POSTER PAPER 8. JONES and KHALID from NAE, Canada and EGGLESTONE from de Haviland compared results obtained by a couple of well known CFD codes (BGK, GRUMFOIL, DRELA, the latter from MIT) with experimental data for four airfoils with different thickness ratios up to 21%. Special emphasis was given to the extraction of wave drag and viscous drag contributions to the total drag both for experimental and theoretical values. Only transition fixed cases were included. Empirical corrections ("Murthy-factor", ΔMach) were found to be necessary in some codes to achieve good agreement with the experiment concerning not only drag. As a consequence the results could not be compared in one graph. Nevertheless a huge amount of data have been presented comparing computed and measured results (viscous drag, wave drag, total drag, Ma_{max}, shock-location and Ma in front of the shock) for each method used. The conclusion is, that further studies are necessary, to reach a satifactory degree of validation of the CFD codes used. Paper relevant to Session II.

POSTER PAPER 9. KESSLER, PERIC and SHEUERER from the University of Erlangen-Nürnberg described a method for the error estimation during numerical flow simulation applied to the prediction of turbulent recirculating flow. The computational method was based on the 2D, incompressible, time-averaged equations for the conservation of mass and momentum. The k-ε model of Launder and Spalding was used for turbulent flow in a finite volume numerical approach. The presentation stressed the most important fact the "Validation" is not only comparing computed with experimental results, it requires an estimation of both the solution errors and the measurement uncertainties. This is specially true for the "validation" of turbulence models and an essential exercise to draw reliable conclusions from the comparisons of numerical and experimental data. The error estimation procedure was demonstrated on the rather complex fully turbulent flow over an obstacle in a plane channel. This contribution was an exellent example for the "Spirit of Validation" outlined by the introductory papers in Session I. The continuation of the presented work to achieve improvements in turbulence modeling is highly recommended.

POSTER PAPER 10. KJELGAARD and SELLERS from NASA Langley discussed results from an experimental investigation of the flowfield over a 75° swept delta wing at HAOA. The investigation includes surface flow visualisation, off-body flow visualization and detailed flowfield surveys for Reynolds numbers between .5 and 1.5 million. The measurements were conducted both with pitot pressure probes and 3-component laser doppler velocimeter in order to find out the accuracy of each instrumentation system. Various algorithms have been used to examine the vorticity field surveys. The presentation of computed 3D Navier-Stokes results was rather limited. Sofar a further excellent data-base (comparable to the International Vortex Flow Experiment) relevant to Session III has been presented. Not discussed was the availability of data for outside NASA/US applications.

POSTER PAPER 11. KUBENDRAN, NASA Langley, SUNG and YANG from DTNSC, Bethseda reported on a basic model to represent the wing-juncture to a fuselage. An unswept wing and a flat plate has been used for experimental studies, and numerical solutions of the incompressible Reynolds-averaged Navier-Stokes equations in combination with the Baldwin-Lomax turbulence model have been obtained. Theory tends to overpredict boundary-layer thickness and the location of the vortex. Even the effect of a leading edge fillet could be demonstrated in both ways to eliminate leading edge flow separation and to reduce drag. The validation of the numerical code was restricted to the comparison of measured and computed data. Paper relevant to Session IV.

POSTER PAPER 12. BORETTI and MARTELLI from the University of Florence reviewed the research on numerical modeling of turbulent reactive gas flows in order to provide improved analytical models for combustion devices. The mathematical approach is based on the solution of the unsteady Navier-Stokes equations, a low Reynolds number k-ε model and a time-marching procedure is applied to reach steady solutions. Both non-reacting and reacting gas flows have been investigated and first results obtained on coarse grids have been compared with experimental data in fair agreement. Further work is needed especially the application to more complex testcases of practical interest.

POSTER PAPER 13. MÜLLER and RIZZI from the FFA, Sweden, contributed to the subject of the solution of the Navier-Stokes equations, this time for laminar compressible flow around delta wings. The geometry of the International Vortex Flow Experiment (see Session III) has been selected at transonic flow condition. The computed results show primary, secondary and even tertiary vortices, the first two in qualitative agreement with the measurements. In comparison with the Euler results presented in Session III, a considerable improvement has been demonstrated even when the laminar N.S. solution is compared with the turbulent wind tunnel experiment. But still needed is the introduction of a model for transition and turbulence.

POSTER PAPER 14. ROSSOW, KROLL, RADESPIEL and SCHERR from the DFVLR in Braunschweig described the problem of accuracy of finite volume methods for solving the unsteady Euler Equations. Cell vertex schemes used for discretization are compared with cell centered schemes concerning requirements found for spatial discretization for first order accuracy on arbitrary meshes. It is demonstrated that for 2D flow problems, methods based on cell vertex scheme establish first order accuracy independent of the smoothness of the grid, whereas methods based on a cell centered schemes do not. In the case of 3D flow analysis on structered meshes, even cell vertex schemes need additional effort in order to establish accuracy. The higher accuracy is shown for the NACA 0012 for c_p (but not for the total pressure losses). This thesis has been discussed extensively during the meeting and was not confirmed up till now by other sources in the literature. Further applications in the paper deal with the flow field in a nozzle and the flow around a powered nacelle. No validation with experiments has been attempted. Paper relevant to VI and VII.

POSTER PAPER 15. HUMMEL from the University of Braunschweig continued his well documented work on separated flow, proposing several experiments as test cases for CFD applications. He concentrated himself in the present paper on low speed flows around delta wings, double delta wings and delta canard configurations as well as on hypersonic flows in axial corners of intersecting wedges. All experimental investigations on separated flows have been carried out at the University of Braunschweig, extensive documentation of the test data is available on request. No computational efforts were reported. Paper relevant to Session III and V.

POSTER PAPER 16. SUTTON, ZOBY and HAMILTON from NASA Langley presented an overview of CFD methods to demonstrate the capabilities of numerical analysis to predict aerothermal environment around a reentry vehicle. Comparisons of computed and measured data are included, mainly based on flight data, to check the real-gas options provided in several codes. The codes investigated range from a direct simulation Monte Carlo Method to investigate the high-altitude, low-density noncontinuum flow to 3D inviscid plus boundary layer methods to investigate the lower altitude, continuum flow condition. The review highlights the prospects and future potential at NASA for prediction of the aerothermal flow around hypersonic flight vehicles. The individual validation process for each computer code was not described in detail but the comparison with experiments presented in the paper was impressive and showed in most cases satisfactorily agreement. This paper should have been included in Session IV.

General Comments on the POSTER SESSION

The experience has shown that all presentations during the Poster Session consist of 15 to 20 Figures, the same amount which could be presented in a 20 minutes speech. The lively discussions with the authors in front of their displays showed, that there was even more active interest expressed by the audience, than observed in the regular sessions, where often less than 5 minutes were left to questions and comments from the floor. As a final recommendation from this exercise, it could be stated, that this first Poster Session within an AGARD FDP Symposium was fully successful and an efficient way to extend the number of possible contributions to a meeting without extending the symposium to a whole week, (39 papers and 19 Poster papers ends up with 58 presentations at this symposium). It stimulates the discussions and it should be repeated every time when a sufficient number of good abstracts have been submitted. The next program committee should consider to extend the time of the exposure, perhaps to the duration of the whole symposium, in order to give to all observers time enough, to study each presentation more in detail and to contact the author individually.

5. CONCLUSIONS

As has been mentioned before several times, this Symposium on **"Validation of Computational Fluid Dynamics"**, has seen many excellent presentations. But at least 5o% of the papers have not been dealing with "Validation" in the sense written down in the Call for Papers by the program committee. They have been commercials for the huge variety of existing computercodes ("Software-Power") in different locations. Each Session of the meeting has been commented allready extensively, so in summary only the most important statements should be repeated :

(1) CFD plays more and more an important **supplementary** role to the experiment in engineering design and analysis.

(2) There is a driving motivation from industry for code validation.

(3) Experimental Data-Bases specially tailored to Code Validation are available and well documented.

(4) AGARD plays a leading role during the last years in initiating international activities in the related field.

(5) Research Institutes working more or less experimentally in the past are now developping and running their own CFD codes.

(6) The experimentalist and the theoretician are comming more and more together.

(7) Numerical and experimental flow simulation are more and more understood as complementary tools, ("no replacement of the Windtunnel").

(8) Good experience with the Poster Session. An efficient tool to extend the content of the meeting.

(9) The process of "Code Validation" is often **misunderstood** as "comparison" of computed and measured Data.

(10) Code validation in a mathematical sense, in terms of stability convergence and uniquness of the solution is was not always transparent.

(11) Comparison with experimental data was often shown for too complex geometries or too complex flow which not always belong to each other.

(12) Dominating flow parameters or sensitivity and individual limitations of the applicability were not always identified.

(13) Data bases for flow code validation have to be specially designed, carfully performed during the wind tunnel tests, and all boundary conditions have to be specified in addition.

(14) The topic of computer cost was not addressed at all. In some cases the cost for extensive calculations reach the level of cost for the experiment.

(15) There is a need of improvement of models to represent transition, separation criteria, separation, turbulence, real gas effects and reacting gas effects in computation.

(16) Main emphasis should be given to **three-dimensional** flow.

(17) Need for validation of experiments also, (identical models in different wind tunnels, "Experimental Benchmarks").

(18) More emphasis should be devoted to understand disagreement between computed and measured results. In some presentations the agreement was to perfect.

(19) In many cases "apples" were compared with "bears", this has been valid for viscid/inviscid or laminar/turbulent comparisons.

This Symposium war a step in the right direction, but perhaps not far enough as could habe been expected by the optimists. So this subject should be addressed again in the near future. There was a statement made by BRADLEY at the beginning of the meeting : **"Nobody believes the result of a CFD calculation except the one who has performed the calculation - but - everybody believes the result of an experiment except the one who dit it !"** After a busy meeting with 53 papers presented, the question may be allowed : who has been converted to a believer ?

APPENDIX A. PROGRAM COMMITTEE

Dr.R.G.BRADLEY, (co-Chaiman)
Director, Aerospace Technology Dept.
GD/FW-MZ 2888
Fort Worth division
P.O.Box 748 Fort Worth, TX 76101 - USA

Dr. W.SCHMIDT (Co-Chairman)
Head of Aerodynamics
Dornier GmbH, BF
Postfach 1420
D-7990 Friedrichshafen - FRG

M.l'Ing. en Chef B.MONNERIE
Directeur Adjoint
Direction aerodynamique
B.P. 72
ONERA
92322 Chatillon - FR

Professor D.PAPAILIOU
Department of Mechanical Engineering
University of Patras
Rio 26001
Patras - GR

Dr.Ing. G.BUCCIANTINI
Aeritaloia-Societa Aerospatiale Italiana
Gruppo Velivoli Combattimento
Corso Marche 41
10146 Torino - IT

Professor Ir.J.W.SLOOFF
National Aerospace Laboratory, NLR
Anthony Fokkerweg 2
1059 Amsterdam - NE

Professor H.NOERSTRUD
Appt. 24
27 rue Noulet
F-31400 Toulouse - FR

Professor A.F.de O.FALCAO
Pavilhao de Marquinas
Instituto Superior Tecnico
1096 Lisboa Codex - PO

Mr. P.R.BIGNELL
BAe PLC, Sowerby Research Cenbre
Naval Weapons Division
FPC 067, PO Box 5
Filton
Bristol BS12 7QW - UK

Dr. R.A.GRAVES
Director, Aerodynamics Div.
NASA/HQ/Mail Code RF
Washington D.C. 20546 - USA

APPENDIX B. LIST OF PAPERS

Session I - CFD VALIDATION CONCEPTS

1. R.C.BRADLEY
"CFD Validation Philosophy"

2. J.G.MARVIN
"Data Accuracy and Benchmark Experiments"

3. J.W.BOERSTOEL
"Numerical Accuracy Assessment"

Session II - EXTERNAL FLOW I : AIRFOILS

4. J.F.FULKNER, D.J.WEEKS, P.R.ASHILL
"Wind Tunnel Experiments on Aerofoil Models for the Assessment of Computational Flow Methods"

5. C.L.RUMSEY, K.W.ANDERSON
"Parametric Study of Grid Size, Time Step, and Turbulence Modeling on Navier-Stokes Computations over Airfoils"

6. J.R.VIEGAS, M.W.RUBESIN, R.W.MACCORMACK
"On the Validation of a Code and a Turbulence Model Appropriate to Circulation Control Airfoils"

7. H.W.STOCK, W.HAASE, H.ECHTLE
"Comparative Study of Calculation Procedures for the Viscous Flow Around Profiles in the Transonic Regime"

8. F.GRASSO
"Numerical Solution of Compressible Navier-Stokes Flows"

Session III - EXTERNAL FLOW II : VORTEX FLOWS

9. A.ELSENAAR, L.HJELMBERG, K.BÜTEFISCH, W.J.BANNINK
"International Vortex Flow Experiment"

10. B.WAGNER, S.M.HITZEL, M.A.SCHMATZ, W.SCHWARZ, A.HILGENSTOCK, S.SCHERR
"Status of CFD Validation on the Vortex Flow Experiment"

11. T.T.NG, R.C.NELSON, F.M.PAYNE
"Flow Field Surveys of Leading Edge Vortex Flows"

12. J.L.FULKNER, P.R.ASHILL
"A Theoretical and Experimental Evaluation of a Numerical Method for Calculating Supersonic Flows over Wing-Body Configurations

Session IV - EXTERNAL FLOW III : WINGS/WING BODY

13. W.KORDULLA, D.SCHWAMBORN, H.SOBIECZKI
"The DFVLR-F5 Wing Experiment - Towards the Validation of the Numerical Simulation of Transonic Viscous Wing Flows"

14. W.K.ANDERSON, J.T.BATINA
"Accurate Solutions, Parameter Studies and Comparisons for the Euler and Potential Flow Equations"

15. M.A.SCHMATZ, A.BRENNEIS, A.EBERLE
"Verification of an Implicit Relaxation Method for Steady and Unsteady Viscous and Inviscid Flow Problems"

16. G.HECKMANN
"Fiabilité et Validité des Codes de C.F.D. Comaparaison au Vol et à la Soufflerie"

17. D.BARBERIS, B.CHANETZ
"Décollement sur Obstacles de Type Ellipsoide - Expériences de Validation et Modélisation"

18. M.P.CARR
"Accuracy Study of Transonic Flow Computaions for Three Dimensional Wings"

19. N.VOOGT, W.J.L.MOL, J.STOUT, D.F.VOLKERS
"CFD Applications in Design and Analysis of the Fokker 50 and Fokker 100"

Session V - EXTERNAL FLOW IV : HIGH SPEED FLOWS

20. G.S.DEIWERT, A.W.STRAWA, S.P.SHARMA, C.PARK
"Experimental Program for Real Gas Flow Code Validatiou at NASA Ames Research Center"

21. M.DORMIEUX, C.MAHE
"Calculs Tridimensionels de l'Interaction d'un Jet Lateral avec un Ecoulement Supersonique Externe"

22. PH.GUILLEN, J.LORDON
"Simulation Numérique d'Ecoulements Supersoniques Décollés autour de Missiles Tactiques"

23. P.A.SHEPHARD, R.G.TOD
"Development and Application of a Weapons Multi-Block Suite"

24. A.D.ZWAN, R.S.CROOKS, W.J.WHATLEY
"Arcjet Validation of Surface Catalycity Using viscous Shock-Layer Approach"

Session VI - INTERNAL FLOW I : TURBOMACHINERY

25. L.A.POVINELLI
"CFD Validation Experiments for Internal Flows"

26. W.KOSCHEL, M.LÖTZERICH, A.VORNBERGER
"Solution of Unstructered Grids for the Euler and Navier-Stokes Equations"

27. F.THIBAUD, A.DROTZ, G.SOTTAS
"Validation of an Euler Code for Hydraulic Turbine"

28. B.LAKSHMINARAYANA, K.R.KIRTLEY, M.WARFIELD
"Computaional Techniques and Validation if 3D Viscous/Turbulent codes for Internal Flows"

29. C.MARESCA, D.FAVIER, M.N.MBA, C.BARBI
"Validation á l'Aide d'Essais en Soufflerie de codes de Calcul du Champ Aérodynamique de Rotors et d'Helices dans des Conditions de Vol Variées"

30. L.E.ERICKSON, J.T.BILLDAL
"Validation of a 3D Euler/Navier-Stokes Finite Volume Solver for a Radial Compressor"

31. H.W.HAPPEL, B.STUBERT
"Computation of 3D Transonic Cascade Flow and Comparisons with Experiments"

Session VII - INTERNAL FLOWS II : INTAKES AND DUCTS

32. H.BUERS, S.LEICHER, P.A.MACKRODT
"Numerical and Experimental Investigation of Engine Inlet Flow with the Dornier EM2 Supersonic Inlet Model"

33. R.G.SEMMERS, D.G.ARBITER, R.G.DYER
"Efforts Towards the Validation of Computional Fluid Dynamic Code for Analysis of Internal Aerodynamics"

34. R.J.KIND, F.M.YOWAKIM, P.M.REDDY
"Measurements and Computaions of Swirling Flow in a Cylindrical Annulus"

35. J.M.M.BARATA, D.F.G.DURAO, M.V.HEITOR, J.J.MCGUIRK
"On the Validation of 3D Numerical Simulations of Turbulent Impinging Jets Through a Crossflow"

36. M.CIOFALO, M.W.COLLINS
"Time-Dependent Numerical Simulation of the Starting Flow of an Incompressible Fluid Past a Downstream-Facing Step"

37. D.H.RUDY, A.KUMAR, J.L.THOMAS, P.A.GNOFFO, S.R.CHAKRAVARTHY
"A Comparative Study and Validation of Upwind and Central-Difference Navier-Stokes Codes for High-Speed Flows"

APPENDIX C. POSTER PAPER SESSION

1. A.AMENDOLA, R.TOGNACCINI, J.W.BOERSTOEL
   "Validation of a Multi-Block Euler Flow Solver with Propeller Slipstream Flows"

2. W.J.BANNINK, E.M.HOUTMAN, S.P.OTTOCHIAN
   "Investigation of the Surface Flow on Conical Bodies at High Subsonic and Supersonic Speeds"

3. B.FAVINI, F.SABETTA, L.ZANNETTI
   "Design and Validation of a @D Euler Code"

4. J.C.F.PEREIRA, F.DURST
   "Finite Difference Methods in Recirculating Flows"

5. M.C.P.FIRMIN, M.A.MCDONALD
   "The Design of the GARTEUR Low-Aspect Ratio Wing for Use in the Validation of Shear Layer and Overall Flow Prediction Methods"

6. J.Y.BALTAR, E.TJONNELAND
   "Slender Cone CFD and Experimental Data Comparisons in Hypersonic Flow"

7. M.FORTIER
   "Validation of a User-Friendly CFD Code for Prediction of the Aerodynamic Characteristics of Flight Vehicles"

8. D.J.JONES, M.KHALID, B.EGGLESTONE
   "Comparison of Theory and Experiment for Four Supercritical, Low Drag Airfoils"

9. R.KESSLER, M.PERIC, G.SCHEURER
   "Solution Error Estimation in the Numerical Predictions of Turbulent Recirculating Flows"

10. S.O.KJELGAAARD, W.L.SELLERS III
    "Detailed Flowfield Measurements over a 75° Swept Delta Wing for Code Validation"

11. L.A.KUBENDRAN, C.H.SUNG, C.I.YANG
    "Experiments and Code Validation for Jucture Flows"

12. A.A.BORETTI, F.G.MARTELLI
    "Accuracy and Efficiency of a Time-Marching Approach for Cumbustor Modeling"

13. B.MÜLLER, A.RIZZI
    "Large-Scale Viscous Simulation of Laminar Vortex Flow over a Delta Wing"

14. C.ROSSOW, N.KROLL, R.RADESPIEL, S.SCHERR
    "Investigation of the Accuracy of Finite Volume Methods for 2- and 3Dimensional Flows"

15. D.HUMMEL
    "Documentation of Separated Flows for Computational Fluid Dynamics Validation"

16. K.SUTTON, E.V.ZOBY, H.H.HAMILTON
    "Overview of CFD Methods and Comparisons with Flight Aerothermal Data"

REPORT DOCUMENTATION PAGE

1. Recipient's Reference	2. Originator's Reference	3. Further Reference	4. Security Classification of Document
	AGARD—AR-257	ISBN 92-835-0504-2	UNCLASSIFIED

5. Originator Advisory Group for Aerospace Research and Development
North Atlantic Treaty Organization
7 rue Ancelle, 92200 Neuilly sur Seine, France

6. Title TECHNICAL EVALUATION REPORT ON THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS

7. Presented at

8. Author(s)/Editor(s)	9. Date
by P.W.Sacher. Edited by R.G.Bradley, Jr and W.Schmidt	May 1989

10. Author's/Editor's Address	11. Pages
See Flyleaf	26

12. Distribution Statement This document is distributed in accordance with AGARD policies and regulations, which are outlined on the Outside Back Covers of all AGARD publications.

13. Keywords/Descriptors

Fluid dynamics	Experimental data
Algorithms	Validity
Computation	Comparison

14. Abstract

This report reviews and evaluates the Fluid Dynamics Panel AGARD Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May, 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes and to ensure that the mathematical and numerical schemes employed in the codes model correctly the critical physics of the flow field under consideration. The evaluator addresses each of the papers presented separately and makes general comments on the seven major topic sessions. In addition, a Poster Session is reviewed in detail.

It is evident that the new possibilities of CFD provide efficient tools for Analysis and Design in the Aeronautical Industry, but it is also evident that in spite of the existence of a number of excellent experimental databases, there is still a need for efforts in validating the computer programs both by experiment as well as by numerical exercises.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP-437 Volume I and are listed in an Appendix to this report. The Poster Papers are listed in the Appendix and published in CP-437 Volume II.

AGARD Advisory Report No. 257
Advisory Group for Aerospace Research and Development, NATO
TECHNICAL EVALUATION REPORT ON THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS
by P.W.Sacher. Edited by R.G.Bradley, Jr and W.Schmidt
Published May 1989
26 pages

This report reviews and evaluates the Fluid Dynamics Panel AGARD Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May, 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes

P.T.O

AGARD-AR-257

Fluid dynamics
Algorithms
Computation
Experimental data
Validity
Comparison

AGARD Advisory Report No. 257
Advisory Group for Aerospace Research and Development, NATO
TECHNICAL EVALUATION REPORT ON THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS
by P.W.Sacher. Edited by R.G.Bradley, Jr and W.Schmidt
Published May 1989
26 pages

This report reviews and evaluates the Fluid Dynamics Panel AGARD Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May, 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes

P.T.O

AGARD-AR-257

Fluid dynamics
Algorithms
Computation
Experimental data
Validity
Comparison

AGARD Advisory Report No. 257
Advisory Group for Aerospace Research and Development, NATO
TECHNICAL EVALUATION REPORT ON THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS
by P.W.Sacher. Edited by R.G.Bradley, Jr and W.Schmidt
Published May 1989
26 pages

This report reviews and evaluates the Fluid Dynamics Panel AGARD Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May, 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes

P.T.O

AGARD-AR-257

Fluid dynamics
Algorithms
Computation
Experimental data
Validity
Comparison

AGARD Advisory Report No. 257
Advisory Group for Aerospace Research and Development, NATO
TECHNICAL EVALUATION REPORT ON THE FLUID DYNAMICS PANEL SYMPOSIUM ON VALIDATION OF COMPUTATIONAL FLUID DYNAMICS
by P.W.Sacher. Edited by R.G.Bradley, Jr and W.Schmidt
Published May 1989
26 pages

This report reviews and evaluates the Fluid Dynamics Panel AGARD Symposium entitled "Validation of Computational Fluid Dynamics" held 2-5 May, 1988 in Lisbon, Portugal. The purpose of the Symposium was to assess the State of the Art of Validation of Computer Codes

P.T.O

AGARD-AR-257

Fluid dynamics
Algorithms
Computation
Experimental data
Validity
Comparison

and to ensure that the mathematical and numerical schemes employed in the codes model correctly the critical physics of the flow field under consideration. The evaluator addresses each of the papers presented separately and makes general comments on the seven major topic sessions. In addition, a Poster Session is reviewed in detail.

It is evident that the new posibilities of CFD provide efficient tools for Analysis and Design in the Aeronautical Industry, but it is also evident that in spite of the existence of a number of excellent experimental data-bases, there is still a need for efforts in validating the computer programs both by experiment as well as by numerical exercises.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP-437 Volume I and are listed in an Appendix to this report. The Poster Papers are listed in the Appendix and published in CP-437 Volume II.

ISBN 92-835-0504-2

and to ensure that the mathematical and numerical schemes employed in the codes model correctly the critical physics of the flow field under consideration. The evaluator addresses each of the papers presented separately and makes general comments on the seven major topic sessions. In addition, a Poster Session is reviewed in detail.

It is evident that the new posibilities of CFD provide efficient tools for Analysis and Design in the Aeronautical Industry, but it is also evident that in spite of the existence of a number of excellent experimental data-bases, there is still a need for efforts in validating the computer programs both by experiment as well as by numerical exercises.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP-437 Volume I and are listed in an Appendix to this report. The Poster Papers are listed in the Appendix and published in CP-437 Volume II.

ISBN 92-835-0504-2

and to ensure that the mathematical and numerical schemes employed in the codes model correctly the critical physics of the flow field under consideration. The evaluator addresses each of the papers presented separately and makes general comments on the seven major topic sessions. In addition, a Poster Session is reviewed in detail.

It is evident that the new posibilities of CFD provide efficient tools for Analysis and Design in the Aeronautical Industry, but it is also evident that in spite of the existence of a number of excellent experimental data-bases, there is still a need for efforts in validating the computer programs both by experiment as well as by numerical exercises.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP-437 Volume I and are listed in an Appendix to this report. The Poster Papers are listed in the Appendix and published in CP-437 Volume II.

ISBN 92-835-0504-2

and to ensure that the mathematical and numerical schemes employed in the codes model correctly the critical physics of the flow field under consideration. The evaluator addresses each of the papers presented separately and makes general comments on the seven major topic sessions. In addition, a Poster Session is reviewed in detail.

It is evident that the new posibilities of CFD provide efficient tools for Analysis and Design in the Aeronautical Industry, but it is also evident that in spite of the existence of a number of excellent experimental data-bases, there is still a need for efforts in validating the computer programs both by experiment as well as by numerical exercises.

The papers presented at the Symposium are published in AGARD Conference Proceedings CP-437 Volume I and are listed in an Appendix to this report. The Poster Papers are listed in the Appendix and published in CP-437 Volume II.

ISBN 92-835-0504-2