Defence Science Journal, Vol. 60, No. 6, November 2010, pp. 567-576

GUEST EDITORIAL

Forty Years of Computational Fluid Dynamics Research in India–Achievements and Issues

Debasis Chakraborty

Defence Research and Development Laboratory, Hyderabad–500 058
E-mail: debasis_cfd@drdl.drdo.in

ABSTRACT

A review of the emergence and maturing of computational fluid dynamics (CFD) research in India over the last four decades is presented. The status of in-house developed CFD codes in various aerospace laboratories and academic institutions in the country is described along with their strengths and weaknesses. Although, some level of maturity is achieved in CFD to address the external flow problems of an aerospace vehicle, the slow growth of indigenous reacting CFD codes forced Indian aerospace industry to depend solely on the commercial software for addressing the internal flow problems related to propulsion and combustion. A brief account of various technical and managerial issues in CFD development is presented. A roadmap is proposed for the graduation of CFD codes, from analysis tool to design tool.

Keywords: Computational fluid dynamics, CFD codes, aerospace vehicle, propulsion, combustion

1. INTRODUCTION

Over the last few decades, computational fluid dynamics (CFD) has developed into a rich and diverse subject and has emerged as a major component of applied and basic fluid dynamic research along with theoretical and experimental studies. Simultaneous development of advanced computers, numerical algorithms, physical and chemical models of flow physics, etc are responsible for the big impact of CFD in solving both basic and applied scientific/engineering problems.

In America and Europe, different numerical algorithms were developed during 1960's and 1970's in the universities and national laboratories for solving fluid flow equations. These methods were applied to solve relatively simple geometries. In the decade of 1980s aerospace industries took major initiatives and a number of industry standard CFD codes were developed to solve the flow field of the complete aerospace vehicle with all its geometrical complexities. Presently, CFD is an integral part of the design process of airframe and engines for all major aerospace companies in the world. While experimental testing will always remain an integral part of the design, CFD is decreasing the dependence on the more expensive, time-consuming experimental testing or rather using experimental work effectively and economically. Advances of CFD tools for commercial airplane development have saved the aircraft companies tens of million of dollars over the past 25 years. Improved turbomachinery and reduced noise of present-day aerospace engines is due to the improvement in the numerical methods, from potential flows in 1970's to the current CFD maturities to solve multi-scale and multi-physics problems.

Significant advances in numerical algorithm and computer architecture and CFD applications have been made in India since the first review on status of CFD by Prahlad[1]. Desai[2] discusses relative roles of CFD and wind-tunnel testing in the development of aircraft and discusses the use of CFD at National Aeronautical Laboratory (NAL), Bengaluru for the development of transport aircraft. Assessment of CFD work in India by Shevare[3] focuses on CFD research.

The present article gives an account on the growth and evolution of CFD in India and discusses various issues which have to be tackled to graduate CFD from an analysis tool to design tool in the country

2. INDIGENOUS COMPUTATIONAL FLUID DYNAMICS CODES IN AEROSPACE INDUSTRY

Aerospace industry is the major driver of the CFD development in the country. In India, numerical studies of fluid flow problems were initiated at IISc, IIT's, national laboratories and different universities in 1970's. Early works were mostly on integral equation methods (Jadavpur University and IIT, Kharagpur), Panel methods (IIT, Kharagpur; NAL, Bengaluru; VSSC, Thiruvanthapuram), Transonic small perturbation equation methods, method of characteristics, Full-potential equation methods (NAL and VSSC), etc. These results have provided good understanding of flow physics in their range of applications.

In 1981, VSSC, Thiruvanthapuram organised an International Workshop on CFD where many CFD stalwarts (Robert MacCormack, J.Ferziger, Urmila Ghia, etc) delivered lectures. This workshop helped the aerospace designer in the country to appreciate the role of CFD in the design process and accelerated the use of CFD in aerospace design in the country. A few indigenous Euler and Navier Stokes Codes were developed at VSSC for analysing the flow field of strap-on separation from core vehicle (space marching Euler solver)[4], nozzle flow field with solid injection thrust vector control (3-D Euler code with Mackcormack's scheme)[5] and stage-separation flow field of multi-stage launch vehicle (Navier Stokes code with FLIC method)[6]. Although, these pioneering applications demonstrated the potential of CFD in solving complex problems in launch vehicle aerodynamics, but these did not become an integral part of the design process. No further follow-up actions were taken to improve the modelling, to validate, to make the codes user-friendly. The code development activities ended with scientific papers that proclaimed, 'Look, what can be done'. As Dr S Vasanth recalls, one of the important CFD exercises that have gone into the design of launch vehicle is the simulation of the effect of aerodynamics of heat shield separation test on ground[7]. The ground test results of heat shield separation were explained through numerical simulation and the safe separation of PSLV heat shield in vacuum was predicted. The designer had taken the CFD results on trust and gone ahead with the launch of PSLV vehicle without conducting any test of PSLV heat shield in vacuum condition. Repeated success of PSLV heat shield separation is the testimony of correctness of the simulation approach. Figure 1 compares the computational and experimental values of internal heat shield pressure during separation test on ground. The major impact of CFD in launch vehicle aerodynamics could be seen with the development of Cartesian-based 3-D Navier Stokes solver PARAS code[8] in VSSC in mid-1990's. The code enabled the use of CFD techniques for analysing practical configurations as the grid generation efforts were highly simplified. The flow pattern in a launch vehicle base region arising due to interaction of multijets and free stream and the base pressure comparison with flight data reproduced from the works by Kumar[9] is shown in Fig. 2. The code was parallelised in multi-node cluster configurations, so that quick results could be obtained as per designer's requirement. Sufficient user friendliness was incorporated to allow the use of this code by any aerospace designer. This code was marketed to different aerospace laboratories

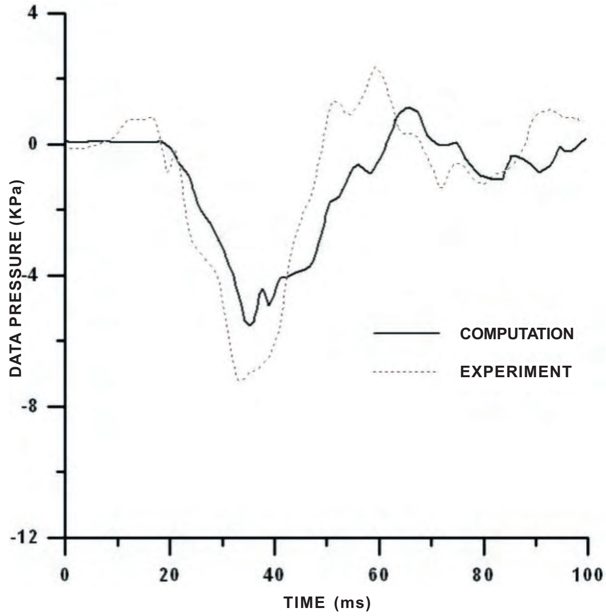
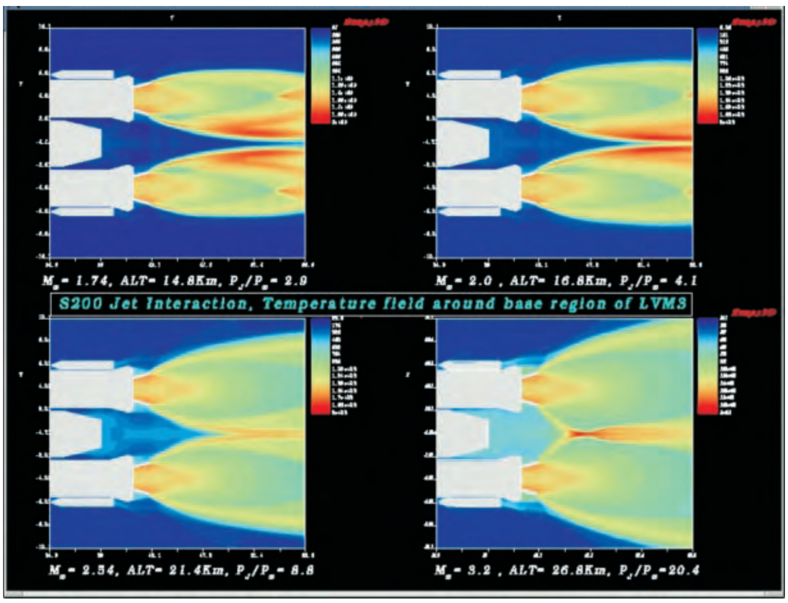
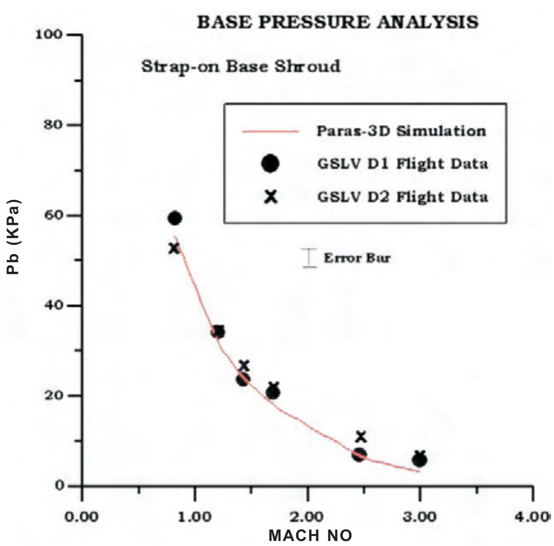


Figure 1. Internal pressure comparison in the heat shield during separation test in ground[7].



(a)



(b)

Figure 2. (a) Flow field in the base region of vehicle with multiple jets, (b) Base pressure comparison between CFD and flight data[9].

in the country and was used extensively to study various flow fields of launch vehicles, fighter aircraft, and missile geometries in the country. VSSC has also developed an indigenous 3-D RANS code UNS3D[10] which is being used for launch vehicle aerodynamics.

In late 1980's, Aeronautical Development Agency (ADA) took a major initiative to unite all the CFD researchers in the country towards analysing the Light Combat Aircraft (LCA) configurations. Number of projects was given to all CFD research centres at IIT's, IISc and these projects were continuously guided and reviewed by ADA designers and senior aerospace faculty. Due to this focused interaction between the designer and CFD researcher led to the development of number of indigenous 3-D Euler and Navier Stokes codes for addressing various aspects of LCA design. The CFD results obtained consequent upon this effort contributed very significantly in LCA aerodynamic design, right from the initial design phase. CNS3D[11] and VISP3D[12] are some of the notable indigenous 3-D RANS codes developed by ADA. These codes were used in analysing various flow problems for the LCA. During 1987–1996, ADA took the leadership of CFD development in the country.

National Aeronautical Laboratory(NAL), Bengaluru developed indigenous industry standard Euler and Navier stokes codes (JEWEL3D[13], JUMBO3D[14], MB-EURANIUM[15]) for aerodynamic characterisation of their in-house aviation projects HANSA and SARAS and also to support different missile and launch vehicle programmes in the country. The viscous flow field of complete SARAS aircraft from JUMBO3D[16] is reproduced in Fig. 3. National Aeronautical Laboratory also contributed significantly in the design and development of wind turbines, underwater bodies, micro air vehicles, etc. through CFD simulations.

In 1990, Dr APJ Abdul Kalam, the then Director, DRDL formed a CFD group to carryout improved missile design through numerical simulation. Algorithmic research in upwind methods and grid-free methods by Prof SM Deshpande and his students at Indian Institute of Science in Bengaluru

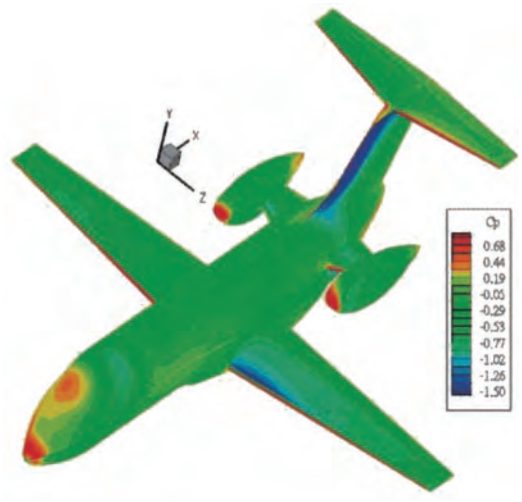


Figure 3. Surface pressure coefficient distribution for SARAS[16], $M_\infty = 0.5$, $\alpha = 5°$, $R_e = 6 \times 10^6$.

were used in developing indigenous three-dimensional Euler codes[17,18] for solving various complex missile aerodynamic problems. Development of industry standard 3-D grid-free method Q-LSKUM[18] has facilitated the solution of many complex problems, like characterisation of control surface deflection, store separation of missiles from fighter aircraft, petal separation at hypersonic Mach number in significant atmosphere, etc. which are not easily amenable to available commercial codes. The development and applications of grid-free method is presented by Dr K Anandhanrayanan in 'Development of an Indigenous 3-D Grid-free Solver and its Applications to Multi-body Aerospace Vehicles' in this volume. State-of-the-art 3-D RANS solver CERANS[19] was developed indigenously at DRDL. Different advanced numerical schemes, physical modelling were incorporated to make the code very robust. Extensive validations were carried out to find its range of applications and error bands. The developed RANS code is routinely applied for characterisation of various missile configurations in complete M-α carpet and to generate important design inputs for many complex missile aerodynamic problems. Lack of user friendliness is one the major impediments for this advanced code to be used by other aerospace designers. The major features of the indigenous 3-D RANS codes in the country have been summarised in Table 1. Roe and AUSM family of schemes are very popular among the aerospace CFD code developers. MS Liou, the developer of AUSM schemes presented 'Evolution of Advection Upstream Splitting Method Schemes' in an article in the same volume. Development and applications of different indigenous CFD codes in aerospace industry is presented in 'Computational Fluid Dynamics in Aerospace Industry in India' by K.P. Singh, *et al.* in this volume.

3. COMPUTATIONAL FLUID DYNAMICS IN ACADEMIA

Academic institutions in the country; mainly IISc and IITs, have made significant contributions towards the development of this emerging technology. Kinetic flux vector splitting and mesh-free methods developed by Prof Deshpande and his students at IISc are put in many in-house developed codes in the aerospace industry. Review of kinetic flux splitting schemes are available[20] and fundamentals and status of mesh-free methods is given in a separate article on Least squares kinetic upwind mesh-free (LSKUM) Method by Prof Deshpande and co-authors in the present issue. Algorithmic research and application initiatives at IISc have resulted in the development of a robust, accurate, scalable unstructured data-based finite volume code HIFUN[21] and was applied to many complex industry standard aerospace problems. The code produced accurate results in many international workshops (AIAA Drag Prediction Workshop, 2009, AIAA HiLift Prediction Workshop 2010). The comparison of computed lift and drag coefficients[22] for high lift NASA trapezoidal wing is shown in Fig. 4. Professor Joseph Mathews and his students have developed a three dimensional RANS code with Roe's scheme and K-ω turbulence

Table 1. Indigenous 3-D RANS codes for industrial applications

Code Name/ organisation	Grid	Numerical schemes	Turbulence models	Applications	Issues
CERANS[17] / DRDL	Structured, unstructured hybrid	KFVS,ROE, AUSM family, HLL etc. Pre-conditioner for low speed flow	Spallart Allamaras (SA), SST (blend of k-ε and k-ω)	Missile geometry in total M-α regime	Not very user-friendly
Hi-FUN[19]/ IISc	Structured, unstructured	KFVS, AUSM family, other upwind methods	SA, SST	Aerospace vehicle	
Zeus[23]/ Zeus Numerix	Structured	Numer of robust schemes	SA	Aerospace vehicles	Efficient pre- and post-processor
Jumbo3D[12] / NAL	Structured	Implicit finite volume	Algebraic models	Aerospace vehicle	Not very user-friendly
MBEURANI-UM[13]/ NAL	Structured	Implicit finite volume	Algebraic models	Aircraft geometry	Not very user-friendly
CNS3D[9]/ADA	Structured	AUSM family, Roe, VanLeer	SA, SST	Aircraft geometry	Not very user-friendly
VISP3D[10]/ADA	Unstructured	Roe, AUSM, Osher	SST	External and internal flows	Not very user-friendly
PARAS[7]/VSSC	Cartesian	Approximate Riemann solver, H2-air laminar chemistry	k-ε	Easy Grid generation for any geometry	Cartesian grid make viscous forces are less accurate
UNS3D[10]/VSSC	Structured	Van Leer	Baldwin-Lomax	Launch vehicle	Not very user-friendly
RANS code[21]/ IISc	Structured	Roe	k-ω	3-D geometry	validation required for industrial use
FEXKER[28]/ VSSC	Structured, unstructured	Finite Element / Finite Volume	k-ε, k-ω, EDC combustion model	Reacting CFD	Not very user-friendly

model[23, 24] which have significant potential for industrial use. Extensive research programmes has been initiated at IISc on large eddy simulations, for both reacting and non-reacting flows. Currently, these studies are confined to study the structures of various canonical, flows, like, compressible round jet and Sandia flame, etc. Prof Joseph Mathew presents the status of 'Large Eddy Simulations' activities in the same volume.

Faculty of Aerospace Engineering of IIT Bombay, Mumbai has made significant CFD contribution on computational electromagnetics, unsteady compressible flow simulation on moving grids, hypersonic flow computations with real gas effects, vortex methods, computational aeroaccoustics, etc. Dr Krishnendu Sinha has reviewed the status of hypersonic aerothermodynamics in an article 'Computational Fluid Dynamics in Hypersonic Aerothermo-dynamics' in the same issue. The development of efficient integrated geometric modeler, grid generator, solver and post-processor led to the development of a good industry standard CFD package IIT-Zeus[25]. This initiative was incubated and has given birth to Zeus-Numerix– the first CFD company in the country with homegrown CFD package. The company is supporting the solution of CFD problems in many industries and is involved in CFD education as well.

Computational fluid dynamics research at IIT Delhi, New Delhi aimed at solving industrial problems through fundamental understanding of flow physics and development of efficient numerical algorithm for incompressible and compressible flows, multi-phase flows, time-periodic and oscillating flows, reacting flows, etc. Significant CFD research is continuing at IIT Kanpur on RANS, LES, and DNS to study various aspects of transition and turbulence, parachute aerodynamics, flow past multi-element aerofoil and wings, flow instability in intakes, aerodynamic shape optimisation, flow in turbomachinery, etc. Figure 5 reproduce the LES results on LPT blades with incoming wakes[26].

The development of second order accurate finite volume solver Aircraft Multiblock Euler Solver (AMES) at IIT Kharagpur has helped ADA to generate aerodynamic loads for various components of LCA aircraft and air data calibration of flow measurement sensors. The code was extended for hypersonic flow using equivalent ν-concept[27]. Simulation of incompressible laminar viscous flow under the action of magnetic forces, aeroaccoustics, simulation of flow-induced vibration, coupled slosh dynamics, wind engineering problems, thunderstorm downburst, LES of turbulent reactive flows are some of the recent CFD research at IIT Kharagpur. Current CFD research at IIT Madras, Chennai includes development of numerical algorithm and turbulence model, unsteady flow simulation, simulation of solid propellant combustion, unsteady aerodynamics of biometric and biological flapping, computational studies of regression rate enhancement in hybrid rocket motors, combustion instability in solid propellant rocket motor and gas turbine combustor, etc. Various CFD studies (both algorithmic and application)

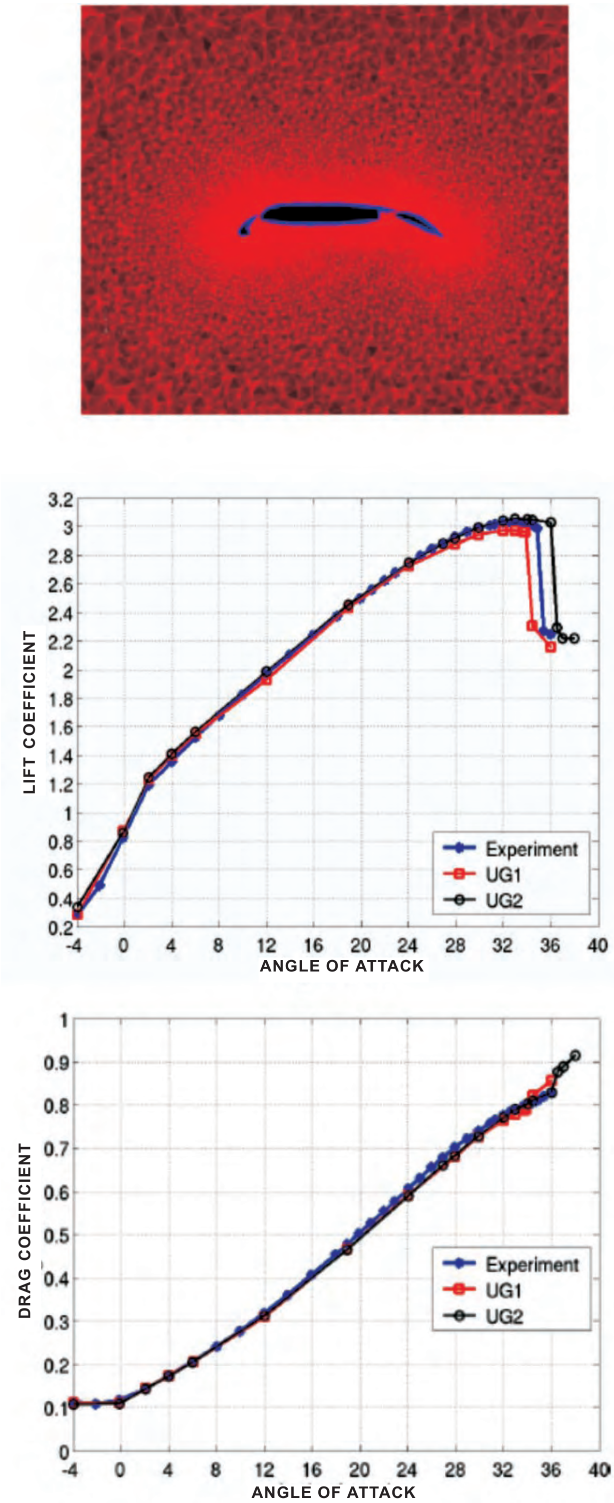


Figure 4. Comparison of computational and experimental values of lift and drag coefficient for NASA trapezoidal wing[22].

being carried out in different universities and other institutes which are not included in the present article.

Aeronautics R&D Board, Defence Research and Development Organisation has supported the development of CFD in a big way by giving unstinted support and

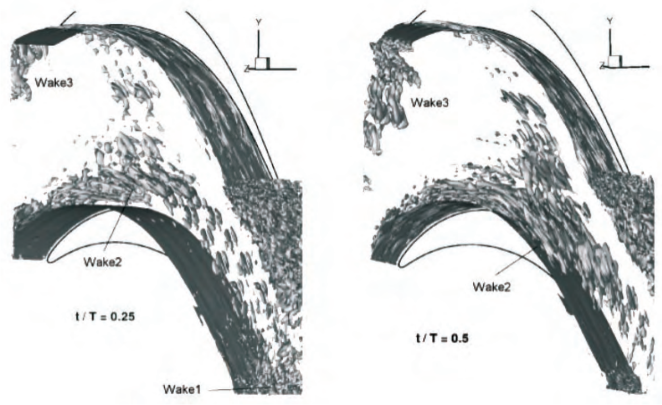


Figure 5. Instantaneous isosurface of vorticity at two time intervals during the wake passing cycle[26].

financial grants to establish six AR&DB CFD centres for CFD research at IISc, Bengaluru; IIT Bombay, Mumbai; IIT, Kanpur; IIT, Madras; IIT, Delhi, and IIT, Kharagpur. A number of novel algorithms were developed in the institutes and was applied to solve practical aerospace problems. An intensive interaction between the faculty and aerospace designers is essential to make these CFD centres effective to serve the aerospace design needs of the country.

4. REACTING COMPUTATIONAL FLUID DYNAMICS

Not much effort was put to develop an industry standard indigenous reacting CFD code in our country. Development of reacting CFD code in academia and various national laboratories focused on the development of various physical and chemical models and testing these models in canonical problems. Most of the aerospace industry depended on commercial CFD codes (CFX, FLUENT, STAR-CD, etc.) for characterisation of different propulsion systems (gas turbine, rocket, ramjet and scramjet) of various aerospace vehicles. Development of reacting CFD code FEXKER[28] in finite element and finite volume framework by Dr T Jayachandran and his group is a notable exception. A more detailed description of the activities of the group is available in the article on ‘Computational Fluid Dynamics in Aerospace Industry’.

I enquired from Prof HS Mukunda as to how things had developed in respect of CFD of internal flow dynamics, and combusting flows in particular, over the last four decades. Responses to this question revealed many aspects of the past. His interest in creating CFD software for combusting flows went back to 1977-78. A team of IISc professors wanted to write a 3-D combustion code to predict the reacting flow field in a combustion chamber so that the CFD and experimental results, namely, the exit temperature profile, could be compared. For reasons of confidentiality, the access to the information on combustor was denied. Propulsion Group at NAL obtained the pheonix code (written by DB Spalding and his company in the UK) from AR&DB Propulsion Panel funds on a commercial basis. The code itself was somewhat complex to use and perhaps had not yet reached maturity for use by anybody other than the code writers themselves. Since only the compiled version of the code was available with possible segments for input of several aspects, it was not easy to use it without the active support of the development team. Not much was accomplished with the use of the code is what we learnt after a period of time. Although, a variety of commercial CFD codes are being used presently to understand the complex flow features inside the *Kaveri* engine being developed at GTRE, the designers in the early years did not have much faith in CFD and organisational support was not provided for the development of indigenous reacting CFD code. During this time, the other engine manufacturers in the world were benefitted immensely from the advancement in CFD technology.

At IISc, a time-dependent, one-dimensional code for flame propagation was developed to deal with complex chemistry as it offered a rich set of aspects to be addressed. This code that received continuous development has been used to address fundamental questions in combustion that could not be resolved with similar software from the USA that was a steady-state calculation approach. The aim then was to expand it to two- and three-dimensions. Further, Prof PJ Paul wrote a two-dimensional reactive flow code, more from research viewpoint. This code has been used for solving two-dimensional propellant combustion issues. The code could not be extended to 3-D problems mainly because of the difficulty for such a development in an academic environment.

In mid 1990s, CFX software was procured for internal flow work at IISc. The software was used both for academic purposes and as well for serious research. This worked on problems that were limited in objectives when it came to understanding complex issues beyond theory or generate results for practical use. Other efforts to progress the use of commercial software for the purpose of understanding the physics on an Institute-wide basis did not mature at all because of a different perspective at the decision-making level. The efforts to block imported software without providing an indigenous alternative prevented progress only for a while. Fluent, CFX, and other software was made available freely to all academic institutions and thus helped wider acceptance of the tool. Various elements of indigenous softwares should be put together to free the reacting CFD from the clutches of 'forbidden for sale' to those who want to use it for nationally important causes. This too requires a path in which individual efforts and their IPR be preserved for getting things put together. Prof PJ Paul presented the status of reacting CFD activities in an article on 'Computational Fluid Dynamics in Combustion' in the same volume.

5. ISSUES

In the early years, all the aerospace laboratories (VSSC, NAL, ADA, and DRDL) adopted the route of indigenous development of the CFD code rather than using the commercial CFD codes available in the market. This was done to have a firm grip on the emerging technology. Also, indigenous development could enable to modify and tailor the codes to suit our own applications and really know where we can use them and where we cannot. Although, a few good solvers were developed, lack of good grid generator for complex configurations and lack of user friendliness remained major bottlenecks for the use of these solvers for industrial applications. Debate continued, whether the commercial CFD software or indigenous development of CFD? Many a times, imported software was opposed without examining if the indigenous development and support was serving the needs of aerospace vehicle designers.

In India, CFD researchers work in isolation. Culture of code sharing has not been developed in the country mainly because of: (i) fear of others using the software benefitting much more without necessary acknowledgement -stated in modern terms, an IPRs (intellectual property rights) issue, and (ii) the fear that others may misuse or criticise the lack of 'goodness' of software. Even within the same organisation, the source codes are not shared and development efforts are duplicated. The development of new capability in a good code always depends on the interest and time available to the code developer. Mostly, codes saw their natural death when the developers cease to be active in the field. Whenever a new CFD development is planned, CFD researchers opt to write the code themselves instead of using an existing one and waste a lot of valuable time in programming, debugging, and validation. Adoption of standard software practices and organisational monitoring is strictly required to develop the teamwork in this important technology.

Recently, to evaluate the accuracy, robustness, efficiency and scalability of indigenous RANS CFD codes in the country, a workshop (SPICES-09) was organised jointly by IISc, CRL (Pune) and ADA (Bengaluru). The NASA Trapezoidal Wing configuration, a High Lift, low aspect ratio wing, for which high quality experimental data is available in open literature, was used for code evaluation. The code evaluation exercise was planned for both structured grid and unstructured grids. Although, 12 participants from various R&D Labs and academic institutions were interested in this code evaluation exercise, only three participants could actually produce results with unstructured grid during the symposium. This exercise has brought out several difficulties, including handling large grid size (which is necessary for high lift computations) and large number of grid blocks. More such code evaluation exercises related to problems of Indian aerospace industry are required to make the indigenous CFD codes to international standard.

Most of the cases, the commercial codes are used as a black box. Not much efforts are put to quantify the uncertainties occurred due to numeric and physical modelling. Detailed validation and verification studies need to be carried out to find the effect of grid, turbulence model, boundary conditions and other physical and chemical models on the solution and CFD researchers must provide the error band of the results and the range of applications to their codes.

Well-instrumented focused experiments should be conducted to anchor various physical and chemical models employed in CFD simulation. Although, CFD could predict the surface pressures and overall forces and moment coefficients reasonably accurately, prediction of wall properties like heat flux rate and skin friction remain little problematic due to proper resolution of wall boundary layer. It is well known that the weakest link in CFD as a scientific discipline is the inadequate description of turbulence. This critical research arena requires long-term vision, sustained support, and rigorous peer review. The pioneering efforts for DNS and LES have opened research avenues for all to follow. More work need to be done to predict flow separation, laminar-to-turbulent transition, combustion, ignition, etc. to predict the finer details of the complex flow field. To analyse a complete aerospace vehicle's performance, it is important to integrate the interaction between aerodynamic, structure, propulsion and control in the iterative design process. An in-depth physical understanding and modelling of these physics requires CFD to expand into interdisciplinary computational domain. Studies on computational aeroelasticity, computational aeroacoustics, computational electromagnetics, computational magneto fluid dynamics, etc. are required for high fidelity multidisciplinary design optimisation to address various design problems encountered in aerospace vehicle. Kim presented 'Computational Elements for High-fidelity Aerodynamic Analysis and Design Optimisation' in a separate article in the same volume.

Although, the commercial codes provided valuable design inputs for many propulsion systems, CFD practitioners face (i) denial of licenses to strategic organisation, (ii) high cost of parallel nodes and annual maintenance contract, and (iii) poor technical support. Incorporation of user-defined models is not very straightforward in these commercial codes. Recent initiative by Scientific Adviser to *Raksha Mantri* and Secretary of Department of Defence R&D for the development of reacting CFD code through national collaboration is noteworthy.

Trained CFD manpower is not adequate to solve different aerospace and non-aerospace problems in the country. About 500 CFD researchers are working in various academic institutes, Government laboratories, and industries. CFD courses are taught at both undergraduate and post- graduate courses in different aerospace and mechanical engineering departments. To enable advanced algorithmic research, mathematics departments of various universities and institutes should introduce CFD syllabus in their curriculum.

The interactions between CFD researchers in the country need to be increased. A two-day Annual CFD Symposium organised by the CFD Division of Aeronautical Society of India is providing a platform to CFD researchers and practitioners across the country for exchange of new ideas of algorithmic research and applications to wide variety of problems. The national conferences for the CFD researchers are summarised in Table 2. The contributory papers should be reviewed vigorously before acceptance to make these conferences truly of international standard. Apart from these conferences, some important international conferences, like Asian CFD Conference (ACFD), are organised in India within a gap of few years on rotation basis. It is necessary to conduct more CFD interactions through focused workshops on specific issues to foster the growth of this subject.

Problems of teraflop computing and terabytes memory, that hindered the growth of CFD in the yester years, do not exist any more. Teraflop computing can be bought, built or hired. Most of the indigenous CFD codes are parallelised in multi-node configurations using open source parallel environment. These codes are scalable and portable to any kind of operating environment and producing the data for complex configurations. The present computing platforms in the country are compared with the leading computing facilities in the world[29] and a wide gap in the capacity is observed. The power of fastest computer in the country EKA at CRL is more than one order magnitude less than the fastest computer in the world. Recent advances in Graphics Processing Unit (GPU) along with CUDA programming should be adopted to achieve higher computing power compared to existing computer environment. To meet the designer's need of total characterisation of aerospace vehicles through numerical simulation, the computing power in the country should be enhanced significantly.

Though CFD has matured significantly, it is still used as an analysis tool. Graduation of CFD code from an analysis tool to design tool is a very challenging process. A five-phase approach has been suggested in the literature[30]. After the development of the demonstrator code, efficient pre-processor and post-processor should be added to produce user-friendly, well-understood and maintainable software. In the next phase, it should enter application research, where the code developers need to work closely with design engineers and project manager. When the

Table 2. Important national conferences for CFD researchers

Conference	Organiser(s)	Frequency
Annual CFD Symposium	CFD Division, Aeronautical Society of India	Annual
Annual General Meeting of AeSI	Aeronautical Society of India	Annual
National convention of Aerospace Engineers	Institute of Engineers	Annual
Symposium on Applied Aerodynamics and Design of Aerospace Vehicles (SAROD)	Trust for Advancement of Aerodynamics in India (TAAI)	Bi-annual
National Conference on Air-breathing Engines and Aerospace Propulsion (NCABE)	NCABE Trust	Bi-annual

Table 3. Major computing platforms in the world

World Rank	Site	Computer (Year/Vendor)	Cores	Speed (Terra Flops)
1	Oak Ridge National Lab. United States	Jaguar - Cray XT5-HE Opteron Six Core 2.6 GHz (2009,Cray Inc.)	224162	1759.00
2	National Supercomputing Centre in Shenzhen (NSCS), China	Nebulae - Dawning TC3600 Blade, Intel X5650, NVidia Tesla C2050 GPU (2010, Dawning)	120640	1271.00
3	DOE/NNSA/LANL United States	Roadrunner - BladeCenter QS22/LS21 Cluster, PowerXCell 8i 3.2 Ghz / Opteron DC 1.8 GHz, Voltaire Infiniband (2009, IBM)	122400	1042.00
4	National Institute for Computational Sciences /Univ. of Tennessee, United States	Kraken XT5 - Cray XT5-HE Opteron Six Core 2.6 GHz (2009,Cray Inc.)	98928	831.70
5	Forschungszentrum Juelich (FZJ) Germany	JUGENE - Blue Gene/P Solution (2009, IBM)	294912	825.50

Table 4. Major computing platforms in India

World Rank	Site	Computer (year/vendor)	Cores	Speed (terra flops)
33	Computational Research Laboratories	'EKA' - 3000 BL460c, Xeon 53xx 3GHz, Infiniband (2008, Hewlett-Packard)	14384	132.80
94	Indian Institute of Tropical Meteorology	Power 575, p6 4.7 GHz, Infiniband (2010, IBM)	3744	55.11
182	Center for Development of Advanced Computing	'PARAM Yuva' Cluster - Xeon 73xx 2.93 Ghz, Infiniband (2008, Hewlett-Packard)	4608	38.10
369	IIT Kanpur	3000 BL460c G6, Xeon X5570 2.93 GHz, Infiniband (2010, Hewlett-Packard)	2944	29.01
389	IT Services Provider (B)	3000 BL460c, Xeon 54xx 3.0GHz, GigEthernet (2009, Hewlett-Packard)	4424	28.36

computational and experimental data do not match, questions whether the assumptions made in the formulation are adequate to explain the flow physics and what additional algorithmic research, geometric pre-processing, physical modelling are required should be asked. Bypass of this stage by taking the code and placing it directly with the designer must be avoided. Designer may not understand underlying theoretical models, algorithm and other numerical issues and may not use the code effectively. When the application experts, code developers, and project managers have together learned the capabilities, limitations, and proper applications of the code, important stand-alone design decision can be taken without supporting experimental comparison and CFD can occupy proper position in the design tool box.

Important CFD developments and applications in design of nuclear reactors, atmospheric sciences (modelling and simulation of monsoon, ocean modelling, thermosolutal convection, tropical cyclone early warning calculation, etc), biomedical research, design of domestic appliances and other non-aerospace applications are not included in the present issue mainly because of the need to have a focused presentation. Majumdar, *et al* presented some applications of indigenous RANS code for low-speed flows including naval hydrodynamics in an article 'Numerical Simulation of Incompressible Turbulent Flow using Linear Eddy Viscosity-based Turbulence Models' in the same issue Various natural disasters like tsunami in Bay of Bengal and Indian ocean, wildfire in Russia, cloudburst in Leh, etc are amenable to simulation through CFD. Efforts should be intensified to simulate such natural calamities to provide important information to the disaster management authorities for better planning for the reduction of their impact.

6. CONCLUSIONS

The growth and evolution of CFD in India for the past forty years have been described. Although, number of indigenous CFD codes with advanced numerical schemes and physical modelling are developed in various aerospace laboratories and academic institutions in the country, these are mostly used to analyse some isolated aerodynamic problems. It is perceived that considerable interaction among the designers, the code developers, and the project managers is required to make these CFD codes a key tool in the design activities. Although, a lot of research continued in various institutes and laboratories to address various advanced topics on combustion, spray dynamics, and multi-phase flows, development of indigenous industry standard reacting CFD code has lagged behind their external flow counterparts. Inadequate organisational support and engine designer's lack of faith on CFD in yester years are responsible for the non-availability of the indigenous reacting CFD code, resulting in continued use of commercial codes for the propulsion characterisation of aerospace vehicles. Although significant progress has been made in the country both in computer power and memory, there exists an order of magnitude gap of computer power between India and other developed countries. CFD has much bigger role to play in the certification of aerodynamic loads, development of low NO_X and low-noise aircraft engine, etc. The major future goal of CFD should be the virtual fly out of aerospace vehicle in supercomputer before the first test flight. The number of CFD researchers in the country should increase significantly to handle the growing need of solution of different kinds of problems involving fluid flow including atmospheric sciences, naval sciences, biomedical research, and prediction of natural disasters, etc.

ACKNOWLEDGEMENTS

Narrating a technical endeavor in a vast country spanning over four decades is really very challenging. The author has immensely benefitted by simulating discussions with many senior teachers, colleagues and friends like Prof HS Mukunda, Dr TS Prahlad, Prof SM Deshpande, Dr S Vasanth, Prof G Shevare, Prof Joseph Mathew, Prof N Balakrishnan, Dr R Krishnamurthy, and many others. The author has also benefitted from the presentations made by various faculty members from IISc and IITs in an AR&DB seminar on CFD during August 2009. The present effort reflects a personal experience in a scientific discipline that is vast; possible omissions may be condoned.

REFERENCES

1. Prahlad, T.S. Status of CFD in India. *In* Proceedings of 14th International Conference on Numerical Methods in Fluid Dynamics, Bengaluru, 11-15 July 1994, pp 39-53.
2. Desai, S. S. Relative roles of CFD and wind-tunnel testing in the development of aircraft. *Current Science*, 2003, **84**(1), 49-64.
3. Shevare, G.R. Assessment of CFD works in India. Science, Technologies and Industry Practice in Aerodynamics and Design. *In* Proceedings of SAROD-2009, Bengaluru, 10-12 December 2009, pp 14-20.
4. Singh K.P. Numerical simulation of inviscid supersonic flow over a launch vehicle with strap-on boosters. Department of Aerospace Engineering, Indian Institute of Science, Bengaluru, 1988. PhD Thesis.
5. Balu, R. Computational studies on three dimensional two phase flows in rocket nozzle thrust vector control. Department of Aerospace Engineering, Indian Institute of Science, Bengaluru, 1989. PhD Thesis.
6. Saxena, S.K. Development and critical assessment of a new FLIC method for base flows. Department of Aerospace Engineering, Indian Institute of Science, Bengaluru, 1987. PhD Thesis.
7. Chakraborty. D. & Vasantha. S. Aerodynamics simulation of heat shield separation test on ground. *Aeronautical Journal*, 1999, **103**(1027), 435-41.
8. Ashok,V.; Kumar, Pradeep; Babu, Thomas C. & Devasia, K.J. Users manual for PARAS-3D. VSSC/ARD/TR-033/2000, 2000.
9. Kumar, Pradeep. CFD and its applications to launch vehicle problems at VSSC. *In* Proceedings of Seventh Asia Computational Fluid Dynamic Conference, Bengaluru Paper No. 23.1, 26-30 November 2007.
10. Unnikrishnan, C. & Balu, R. Development of an unsteady Navier-Stokes solver for aerospace applications. VSSC/ARD/TR/355/95.
11. Prakash, Satya. Building blocks of CNS3D: A compressible Navier-Stokes solver. Report No.ADA/TD/CFD/TR/115, July 2003.
12. Amaldas, J. Richard. Anisotropic tetrahedral mesh generation and flow solution for Navier Stokes equation. Report No. ADA/TD/CFD/TR/104. 2002.
13. Chakrabarty, S.K.; Dhanalakshmi K. & Mathur, J.S. Computation of three dimensional transonic flow using a cell vertex finite volume method for the Euler equations. *Acta Mechanica*, 1996, **115**, 161-77.
14. Chakrabarty, S.K.; Dhanalakshmi K. & Mathur, J.S. Computation of three dimensional transonic viscous flow using JUMBO3D code. *Acta Mechanica*, 1996, **119**, 181-197.
15. Nair, Manoj T.; Rampurawala, Abdul M. & S.K. Saxena. MB-EURANIUM: User's manual. Project Document CF 0110, National Aerospace Laboratories, Bengaluru. 2001.
16. Mathur, J.S.; Dhanalakshmi, K.; Ramesh, V. & Chakrabartty, S.K. Aerodynamic design analysis of SARAS aircraft. *In* recent trends in aerospace design and optimization, *In* Proceedings of SAROD-2005, Hyderabad, December, 2005.
17. Krishnamurthy, R. Kinetic Flux vector splitting scheme for unsteady Euler equations on Moving Grids (KFMG). Aerospace Engineering Department, Indian Institute of Science, Bengaluru, 2002. PhD Thesis.
18. Anandhanarayanan, K. Development and applications of a grid free kinetic upwind solver to multibody Configurations. Aerospace Engineering Department, Indian Institute of Science Bengaluru, 2003. PhD Thesis.
19. Balasubraminan, R.; Anandhanarayanan, K. & Balakrishnan, N. Development of 3-D compressible Reynolds averaged Navier-Stokes solver. ECCOMAS-2004, Jyvaskyla, Finland, July 2004.
20. Deshpande, S.M. Kinetic flux splitting schemes. *In* CFD review: A state-of-the-art reference to the latest developments in CFD, *edited by* M. M. Hafez and K. Oshima, Wiley, New York, 1995.
21. Shende, N. & Balakrishnan, N. HIFUN-3D-1: Users manual. Department of Aerospace Engineering, Fluid mechanics Report. 2004 FM 10, Indian Institute of Science, Bengaluru, India, June 2004.
22. Ravindra, K.; Shende, Nikhil Vijay & Balakrishnan, N. Performance of code HIFUN in SPICES 09. *In* Proceedings of 11th AeSI Annual CFD symposium, Indian Institute of Science, Bengaluru, 11-12 August 2009,
23. Sriram, A.T. Numerical simulations of transverse injection of plane and circular sonic jets into turbulent supersonic

cross flows. Department of Aerospace Engineering, Indian Institute of Science, Bengaluru, 2003. PhD Thesis.
24. Sriram, A.T. and Mathew, Joseph. Numerical simulation of transverse injection of circular jets into turbulent supersonic streams. *J. Propulsion Power*, 2008, **24**(1), 45-54.
25. Jain, Sourabh & Rai, Prashant. Implementation of implicit LUSGS scheme with dual time stepping for 3-D unsteady compressible turbulent flows. *In* Proceedings of 10th AeSI Annual CFD symposium, Indian Institute of Science, Bengaluru, 11-12 August 2008.
26. Sarkar, S. Large Eddy Simulation of some flows of engineering interest. *In* Proceedings of 11th AeSI Annual CFD symposium, Indian Institute of Science, Bengaluru, 11-12 August 2009.
27. Sinhamahapatra, K.P. Unsteady aerodynamics calculations using three dimensional Euler equations on unstructured dynamic grids. *Aeronautical Journal*, 2002, **106**(1059), 269-76.
28. Jayachandran, T. Simulation of turbulent reacting two phase flows in propulsion systems. *In* Proceedings of International Workshop on Turbulent Reacting Flows, IIT Madras, Chennai, 19-20 Dec 2002, pp. 135-52.
29. http://www.top500.org (Accessed on June, 2010)
30. Johnson, F.T.; Tinoco, E.N. & Yu, N.J. Thirty years of development and application of CFD at Boeing commercial airplanes, Seattle. AIAA paper 2003-3439, 2003.

Contributor

Dr Debasis Chakraborty obtained his PhD in Aerospace Engineering from Indian Institute of Science (IISc), Bengaluru. Presently, he is working as Technology Director, Computational Dynamics Directorate, DRDL, Hyderabad. His research interests are CFD, aerodynamics, high-speed combustion, and propulsion. He has about 30 journal and 40 conference publications to his credit.