CURRENT CHALLENGES IN COMPUTATIONAL FLUID DYNAMICS WITH REGARD TO ROCKET ENGINE THRUST CHAMBER SIMULATION

Karol Swiderski
Institute of Aviation

Abstract

Aerospace industry is the first and most prevalent in the use of numerical techniques. It is worth mentioning that the beginning of CFD is dated for early 1960's and the first successes came to prominence in the 1970's. Creation of the CFD-service industry started in the 1980's and its significant expansion took place in the 1990's. In most phases of the development process the aerospace industry was driving CFD to answer to its needs.

In the past decade Computational Fluid Dynamics (CFD) became a common tool in applied aerospace where many different numerical techniques are currently used. The main areas are in the geometrical definition (CAD) of the model which is to be analysed, the computational solution of the flow field (Mesh generation and CFD) including grid adaptation which may be further combined with other disciplines like shape optimisation, structural analysis and so forth. The scientific area of CFD includes technical advances for accurate numerical methods for resolving flow phenomena and realistic physical modeling of the flow itself. CFD can be defined as the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. In all cases the key factor is the discrete continuum representation.

With regard to rocket applications modeling and numerical simulation allows qualitative and/or quantitative analysis of processes difficult to analyse experimentally. This paper presents useful hints to handle complex flows and their setup for a successful solution and the mathematical basis behind the industrial application which allow further developments for rocket applications.

INTRODUCTION

The history of CFD started in the period of 1965-1975. Around that time, it became a combination of physics, numerical mathematics, and, to some extent, computer sciences employed to simulate fluid flows. The first application of the CFD methods was the simulation of transonic flows based on the solution of the non-linear potential equation [1; 2; 3]. With the beginning of the 1980's, the solution of first two-dimensional (2-D) and later also three-dimensional (3-D) Euler equations became feasible. Thanks to the rapidly increasing speed of supercomputers and due to the development of a variety of numerical acceleration techniques like multigrid, it was possible to compute inviscid flows. With the mid 1980's, the focus started to shift to the significantly more demanding simulation of viscous flows governed by the Navier-Stokes equations. CFD has now matured to the point at which it is widely accepted as a key tool for complex design such as rocket. The complexity and range of phenomena is well illustrated in [4].

Rocket propulsion CFD has, in general, lagged behind aeronautical applications. Complexity of the flow physics and hardware geometry involved in rocket engines probably delayed the application of CFD to this area. One of the most significant applications of CFD simulation to rocket engines began in the early 1980's, when NASA carried out a series of upgrades to the Space Shuttle main engine (SSME), developed in the 1970's.

Algorithms have been the subject of intensive development for the past two decades. The principles underlying the design and implementation of robust schemes which can accurately resolve shock waves and contact discontinuities in compressible flows are now quite well established. Despite this, CFD is still not being exploited as effectively as one would like in the design process. This is partially due to the long set-up times and high costs, both human and computational, associated with complex flow simulations. The essential requirements for industrial use are:

- assured accuracy
- acceptable computational and human costs
- fast turn around.

Improvements are still needed in all three areas. Several routes are available toward the reduction of computational costs, including the reduction of mesh requirements by use of higher order discretization schemes, improved convergence to a steady state by sophisticated acceleration methods, fast inversion methods for implicit schemes, and the exploitation of massively parallel computers.

DESIGN PROCESS AND CFD

The design process can generally be divided into three phases: conceptual design, preliminary design, and final detailed design. Improvements in CFD which would allow the elimination of a major cycle in preliminary design would significantly shorten the overall design process and therefore reduce costs.

Many critical phenomena of fluid flow, such as shock waves and turbulence, are essentially nonlinear. They also exhibit extreme disparities of scales. Moreover computational costs vary drastically with the choice of mathematical model. The computational simulation of fluid flow presents a number of severe challenges for algorithm design. At the level of inviscid modeling, the inherent nonlinearity of the fluid flow equations leads to the formation of singularities such as shock waves and contact discontinuities. When viscous effects are also included in the simulation, the extreme difference of the scales in the viscous boundary layer and the outer flow, which is essentially inviscid, is another source of difficulty, forcing the use of meshes with extreme variations in mesh interval. For these reasons CFD, has been a driving force for the development of numerical algorithms.

The algorithm designer faces a number of critical decisions. The first choice that must be made is the nature of the mesh used to divide the flow field into discrete subdomains. The Cartesian mesh minimises the complexity of the algorithm at interior points and facilitates the use of high order discretization procedures, at the expense of greater complexity, and possibly a loss of accuracy, in the treatment of boundary conditions at curved surfaces. This difficulty may be alleviated by using mesh refinement procedures near the surface. With their aid, schemes which use Cartesian meshes have recently been developed to treat very complex configurations [5; 6].

Rocket engine thrust chamber design provides many challenges:

- spray combustion process encompasses many physical processes of different types, temporal and spatial scales, tightly coupled to each other (two-phase mixing, combustion, turbulence, kinetics, instabilities) making environment definition and modeling very difficult
- extremely high heat loads require exotic materials and active cooling
- transient phenomena can lead to severe instabilities in the combustor and nozzle
- performance and durability issues require trade-offs and limit the design envelope

- test data are difficult and very expensive to obtain.

Engineering challenges and technology needs include:
- ability to rapidly predict the flow and thermal environment in the combustion chamber (another CFD challenge – dynamic grid adaptation; highly efficient solvers for 3D transient mixed flows; workable turbulence models; mechanistic combustion models and correlations; PC-based parallel computing capability and large database management and postprocessing technologies)
- interfacing CFD predicted flow and thermal environments and loads with stress and structural dynamics codes and material databases to select materials and coatings, yield and failure limits, active cooling requirements, and margins of safety (triple challenge – CFD has to provide what material scientists and stress engineers need, and they have to understand CFD's limitations and areas of uncertainty)
- chamber durability and injector performance
- combustion stability especially with hydrocarbon based fuels
- acquiring safety test data and experiments to validate models and predictions.

Computational fluid dynamics analyses have been successfully applied in areas related to the prediction and simulation of combustion flow behavior and heat transfer to the internal walls of rocket engine injectors, combustion chambers, and nozzles. These analyses have been used to optimize nozzle entrance geometries, evaluate new step nozzle exit configurations that adapt to altitude changes, determine pressure and temperature profiles in rocket engine chambers and nozzles, and to study the effects of coolant flows in liquid rocket engine chambers on internal walls. These analytical procedures have helped to evaluate anomalies discovered in actual engine firings and to design reliable combustion chamber, nozzle, and coolant arrangements that result in high thrust coefficients under various atmospheric and space conditions. Computational fluid dynamics simulations have also been useful in determining pressure, heating, and insulation requirements for launch vehicles during liftoff, ascent, and reentry into the atmosphere.

CHALLENGES AND POSSIBILITIES

CFD capabilities have been advanced along with computational technologies in general. Many fluid engineering problems can now be simulated, however these are mostly at a single-component level. For instance, it is possible to generate solutions to problems like a combustion chamber or turbopump. To realize the full benefits of CFD, more inclusive modeling will be required, such as systems of pumps; in general coupling of subsystems which are dependent on each other. Attempts to solve these types of problems have been made with some qualitative successes. However, the predictive capability is still very limited, and prediction with accurate physics is yet to be accomplished. This will require inclusion of not only fluid dynamics modeling but modeling of other quantities like thermal loading, turbulence and transition prediction, droplet formation, separation and cavitation physics.

Future development of rockets will rely heavily on system level thinking, robust design principles, multidisciplinary analysis and optimisation, and on the use of high fidelity predictive tools even in the conceptual design cycle. Development testing will be significantly reduced in favour of large scale, high fidelity simulations and virtual engineering practices.

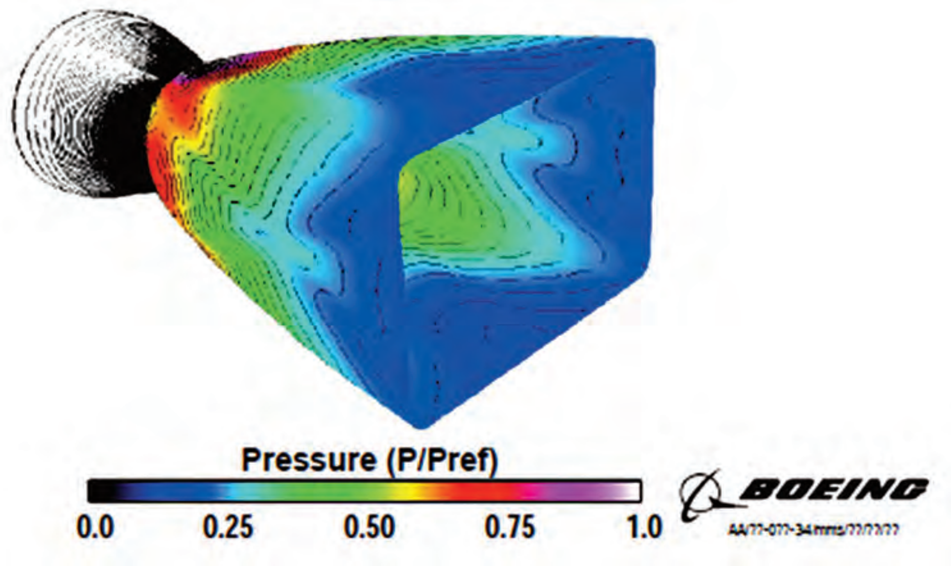

Fig. 3. X-33 thruster analysis [7]

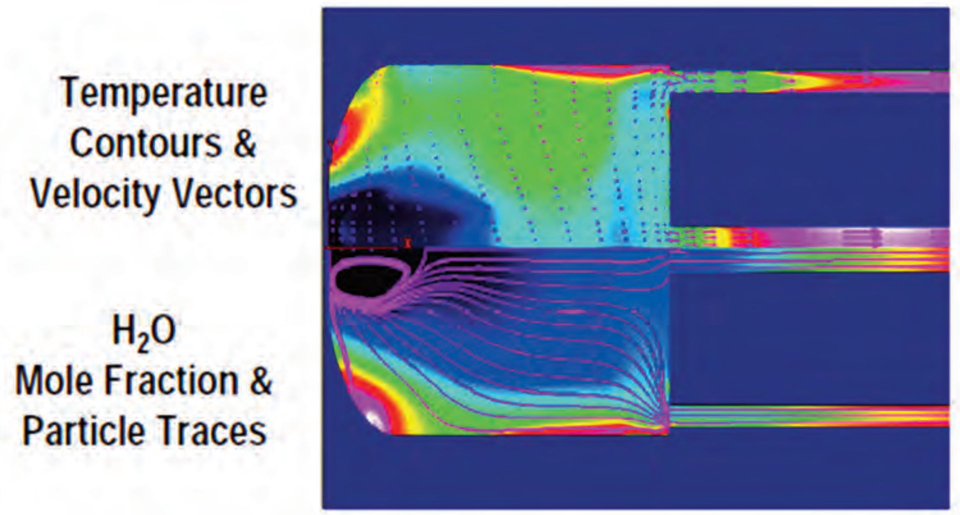

Fig. 4. Hybrid engine combustion chamber [7]

REFERENCES

[1] **E.M. Murman, J.D. Cole.:** *Calculation of plane steady transonic flows*. AIAA Journal. 1971.
[2] **Jameson, A.:** *Iterative solution of transonic flows over airfoils and wings, including flows at Mach 1*. AIAA 2nd CFD Conference. 1975.
[3] **Eberle, A.:** *A finite volume method for calculating transonic potential flow around wings from the minimum pressure integral*. NASA TM 75324. 1978.
[4] **Dyke, M. Van.:** *An album of fluid motion*. The parabolic press, Stanford. 1982.
[5] **J.E. Melton, S.A. Pandya, J.L. Steger.:** *3D Euler flow solutions using unstructured Cartesian and prismatic grids*. AIAA Paper 93-0331. 1993.
[6] **M. Berger, R.J. LeVeque.:** *An adaptive Cartesian mesh algorithm for the Euler equations in arbitrary geometries*. AIAA Paper 89-1930. 1989.
[7] **Boeing.** Overwiew of the state-of-the-practice Liquid Propellant Rocket Engine Design, Analysis, and Test. *AFOSR Conference.* 2004..

Karol Swiderski

AKTUALNE WYZWANIA DLA KOMPUTEROWEJ MECHANIKI PŁYNÓW W ODNIESIENIU DO SYMULACJI KOMORY SPALANIA I DYSZY SILNIKA RAKIETOWEGO

Streszczenie

Przemysł lotniczy przoduje w zastosowaniach i najszerzej wykorzystuje techniki numeryczne. Warto wspomnieć, że początki metody CFD datowane są na początek roku 1960, a jej pierwsze sukcesy zostały wyeksponowane w roku 1970. Tworzenie branży usług CFD rozpoczęło się w roku 1980 a jej znaczne rozszerzenie miało miejsce w roku 1990. W większości etapów rozwoju branży CFD przemysł lotniczy zmuszał ją do udzielania odpowiedzi na swoje pytania.

W ostatnim dziesięcioleciu Computational Fluid Dynamics (CFD) stała się powszechnym narzędziem w obszarze kosmonautyki stosowanej, gdzie obecnie zastosowanie znajdują różne techniki obliczeń numerycznych. Główne obszary zastosowania metody to definicja geometrii analizowanego modelu (CAD), rozwiązania obliczeniowe w polu przepływu (generowanie siatki i CFD), w tym dostosowywanie siatki, które może być połączone dalej z innymi dyscyplinami, takimi jak optymalizacja kształtu, analiza strukturalna, itd. Obszar zastosowań naukowych CFD obejmuje rozwój dokładnych metod numerycznych do rozwiązywania zjawisk przepływu i realistyczne modelowanie fizyczne samego przepływu. Metodę CFD można określić jako analizę systemów przeprowadzaną za pomocą symulacji komputerowych z udziałem przepływu płynów, wymiany ciepła i związanych z nimi zjawisk, takich jak np. reakcje chemiczne. We wszystkich przypadkach, kluczowym czynnikiem jest dyskretna reprezentacja continuum.

W odniesieniu do zastosowań techniki rakietowej, numeryczne modelowanie i symulacja pozwalają na jakościową i/lub ilościową analizę procesów, które trudno jest analizować doświadczalnie. W pracy przedstawiono wskazówki przydatne do modelowania skomplikowanych przepływów oraz ich konfigurowania, które zapewnią pomyślne ich rozwiązywanie oraz matematyczne podstawy dla zastosowań przemysłowych, umożliwiające dalszy rozwój metody dla zastosowań rakietowych.