PROGRESS AND FUTURE PROSPECTS OF CFD IN AEROSPACE : OBSERVATIONS FROM 30 YEARS RESEARCH

Kozo Fujii*
***Institute of Space and Astronautical Science (ISAS)**
Japan Aerospace Exploration Agency (JAXA)
Sagamihara, Kanagawa 229-8510, JAPAN

Keywords: *Computational Fluid Dynamics*

Abstract

Historical perspective of CFD in aerospace in last 30 years is firstly given. It is shown that there remain problems that are geometrically simple but difficult to simulate even though we see a lot of simulations over complex body configurations. The result indicates that CFD research is now in the "specific phase" and requires some innovation.

The innovation includes "evolutional effort" and "revolution". As an example of evolutional effort, LES (Large Eddy Simulaitons)/RANS (Reynolds-averaged Navier-Stokes simulations) hybrid method is taken as an example. Shift from RANS to LES/RANS hybrid occurs not only because of the progress of computers but also because of our recognition that separated flows are inherently unsteady and successful simulations require LES-like computations.

Comment is given that there may be other types of research necessary for CFD to become a real useful tool for the design in addition to just showing CFD capability for complex body configurations. As one of the examples, construction of CFD database is presented. Another issue was to make a CFD infrastructure so that people outside CFD community may use CFD as a tool to hit or refine their ideas.

It is concluded that we have not found the clear clue for the revolution of CFD research but that may come out from the requirement by the design and development process.

1 INTRODUCTION

Computational Fluid Dynamics (CFD) has made great progress in last 30 years. There are numbers of commercial software available now and anyone seems to be able to carry out sophisticated flow simulations even on PC's. When glancing back the history of CFD progress, there were a few epoch-making stages. In the aerospace community, CFD first attracted people's attention in 1970's. The target was simulations of transonic flows. Embedded shock wave was automatically captured and design process of commercial aircraft was drastically changed since then[1]. The equations to be solved have changed from potential to Navier-Stokes equations in early 80's for research applications. In the middle of 80's, CFD in aerospace again attracted people's attention. The topic was simulation of hypersonic flows associated with space transportation system development including reentry vehicles. Aerodynamic heating was a main topic and the solution methods for fluids-chemical reaction were discussed. TVD schemes that create monotone shock structures without parameter tunings became the main solution methods, as unphysical chemical reactions were avoided. Since then, we do not see such epoch-making topics in the aerospace CFD. At first glance, it looks disappointing, but it indicates that CFD technology has developed to a certain level and people use CFD as one of the essential analysis tools even with the remaining problems such as choosing an appropriate turbulence model. The fact that many workshops were held for the discussion on validations and verifications of CFD shows that there are strong needs for CFD as a practical analysis and design tool.

Computer progress has been a strong acceleration factor during the CFD development since supercomputers appeared late 1970's. Figure 1 shows the progress of computers. In a few years, performance of leading-edge computers would be on the order of Petaflops which is roughly ten-millions times faster than the first commercial supercomputer CRAY 1 that appeared late 1970's.

There was a workshop on "computer requirements for computational aerodynamics" at NASA Ames Research Center on 1977[2]. Dean Chapman wrote, in his "Opening Remarks" that there are two major motivations behind CFD and it would not change in coming decades. Two motivations were (1) providing an important new technology capability and (2) economics.

To illustrate the first, he compared wind tunnel experiment and CFD. He insisted as follows. There are many restrictions in the wind-tunnel experiment such as scale effects, wall and support interference, aerodynamic distortion, and else. The restriction of CFD comes from the speed and storage, but the technical trend shows that such limitations are rapidly decreasing.

The second message given by Dean Chapman in 1977 was essentially correct and that happened on CFD for the 30 years since then. Even three-dimensional Navier-Stokes simulations over a 3-D wing is not a difficult task and can be carried out with less than one hour on PC's now as will be discussed later. On the other hand, it is not clear if CFD has shown the expected capability although some of them realized. For instance, it is questionable if we can evaluate scale effect (Reynolds number effect) so far as we use RANS (Reynolds-aerage Navier-Stokes equations) simulations with turbulence models. Progress of computer speeds has not yet solved this problem.

In the present paper, we glance back the development of CFD in last 30 years and see some of the problems we currently have based on the historical perspective. The future direction of CFD is then discussed.

Before closing an Introduction, the author would like to leave the following message. Please remember that the CFD history considered here is limited to the area of practical applications. Everybody knows that there was much earlier effort in 1900's. Also note that the messages below only reflect author's narrow experience, and the examples in this manuscript are taken based on the limited exposure of the author's.

2 Historical Perspective of CFD in Aerospace

Practical flow simulations in Aerospace using Compressible Navier-Stokes equations first appeared in 1985. As shown in the overview article in Aerospace America in 1986[3], transonic flow simulations over a commercial-type wing and a wing-fuselage were carried out in Japan almost at the same time as those for fighter aircraft at NASA Ames Research Center. The results[4,5] are shown in Figs. 2(a) and 2(b). These were the first example of the Japanese GFLOPS supercomputer, FUJITSU VP400 that appeared in 1985. It took about two hours of computer time for the rough convergence to steady state. How fast is current PC's that we are using today? It takes 3-4 μsec per grid point per iterations for our CFD program to simulate three-dimensional compressible Navier-Stokes equations, whereas it took 7-9 μsec on the VPP400 in 1985. As shown in this example, steady-state flow simulations with less than one million grid points may be carried out within an hour or so using current PC's. Although there may be enormous number of cases required for the design process, they may be carried out under reasonable time frame.

Geometry complexity is not the main program for the computer time. People know that bottleneck of the time required for the flow simulation now is not the computer time but the time required for preparing the surface and volume grid data from the CAD geometry. There still remain another problems, however in the current CFD.

Earll Murman, Professor of MIT, gave a general lecture in the ICAS 2000 symposium held in Harrogate, England, and talked about Aeronautical Design engineering and Manufacturing[6]. One of the figures in his manuscript is re-plotted in Fig. 3. The figure showed time evolution of the number of major US aerospace companies. He mentioned that the trend follows a classic pattern of product evolution exhibited by many industries as studied and reported by Utterback. I just take his words here, "In the early years of a new product, the fluid phase, the basic product features are evolving and many startup companies enter the field. At some point, a dominant design emerges when the basic product features become established and a transitional phase is entered. Many factors come into play to establish the dominant design including technology, infrastructure, customer expectations, individual entrepreneurs, etc. At this point, more companies start to leave than enter the field. Innovation starts to switch from product to process technologies, i.e. to design, development, manufacturing innovation. As the product features stabilize the specific phase is reached where significant changes in product features are unlikely". When he listened to Prof. Murman's talk, the author thought that the Utterback's theory also applied to the CFD in aerospace. In early days, we developed numerical algorithms to solve the basic equations, for instance, Non-linear Potential, Euler or Navier-Stokes equations and developed computer programs to conduct the flow simulations. A lot of ideas appeared, such as Implicit schemes; Approximate Factorization, Approximate LU decomposition, LU-SGS, TVD-like schemes; Flux Vector splitting, Flux difference splitting, AUSM, and else. With the aid of such algorithm development and appearance of supercomputers, a lot of researchers entered the area and CFD technology rapidly progressed. It was a grow-up period of CFD and people enjoyed visualized images that showed capability of CFD to handle complex body configurations and complex physics. The period is considered to be "Fluid Phase" where product innovation occurs. There are still some efforts even now but I would say that importance of such effort mainly finished early in 1990's, when we obtained basic CFD methods to solve wide variety of aerodynamic problems (Note that we only discuss limited area of CFD applications. There are areas in which even the mathematical model has not been well established). From the late 80' to middle of 90's, there was a discussion on 'Overset or Patched Structured grid' or 'Unstructured grid', which was the strategy to solve problems. Improvement of the efficiency of CFD solution process using parallel computers was another topic, and even the international symposium named "Parallel CFD" appeared. In this period, small but inevitable effort to use CFD technology for practical problems was the main focus. I call it "transitional phase" where process innovation (although original meaning is totally different). We do not see epoch-making topics in recent years although interdisciplinary applications or design optimization appeared and they use CFD as a one of the solution elements. From one viewpoint, it indicates that CFD technology has developed to certain level where people use CFD as one of the essential analysis tools even with the remaining problems such as choosing an appropriate turbulence model. From the transitional phase to specific phase, the number of researchers entering into the area becomes less than the number of researchers leaving the area. I consider that aerospace CFD technology is now in the "Specific Phase". Change in the product features is unlikely and some innovation is required. Otherwise, the CFD technology just stops there and will not extend it's use.

3 CFD in the Specific Phase

In this section, we look at the current status of CFD.

3.1 CFD .vs. FED

In the past, we, CFD researchers have been trying to prove the capability of CFD. In general, our effort mainly focused to fulfill the requirement to solve practical problems with certain level of fidelity within reasonable time. As a result, we have been showing simulation examples for more and more complex body configurations. CFD research is not completed and it is true that we still have to keep doing it, but at the same time, we need to consider what is required from the people outside the CFD community. There may be other types of research necessary for CFD to become a real useful tool for the design.

Table 1 shows the comparison of CFD with EFD (Experimental Fluid Dynamics) as a wind tunnel experiment to be a representative. This classification may be personally biased and there may be a much different way of thinking, but please just take this as one idea. The last column is the weak point of each. In the wind tunnel experiment, many people do not spend much time for the items described in the table. There are people conducting research for new measurement techniques such as PIV or PSP, but majority of people simply use the existing techniques. They do not develop pressure sensors, force balances for their experiments. In other words, the items for EFD shown in Table 1 are not the main topics for the experimental research but just tools for their experiments. On the other hand, when talking about CFD, people imagine the research shown in Table 1 and actually the conferences on CFD have focused on these topics. CFD has been focusing too much on the development as a computational tool and the view from aerospace engineering has been somewhat lost in the mind of CFD researchers. We will come back to this point in the Section 4 again.

3.2 Simple Problems Still Remain

CFD researchers know that CFD is a powerful tool but not a almighty tool. Even though there appear a lot of simulation examples for complex body configurations, there left many physical problems that look simple but difficult to simulate. Computations may be easy but simulations (meaning with satisfaction to the required accuracy) is difficult. Here, three examples are shown for the future discussion of the present paper.

3.2.1 Thin-airfoil Stall Characteristics

Precise estimation of maximum lift and stall angle of a wing is an important issue for the aerodynamic design of aircraft. It is important and necessary to develop a prediction method of such unsteady flows at high Reynolds number within practical computational costs. Paul Rubbert, the leader of CFD group at Boeing Commercial Company from 80's until he retired, said to us "Flow simulations at cruise condition can be done by non-linear potential equations and boundary layer theory. Solutions of Navier-Stokes equations are needed for the simulation under buffet or stall conditions" when he visited Japan in 1985. CFD has not yet answered his comments even now. It is still difficult to simulate massively-separated unsteady turbulent flows at high angles of attack near stall even though the conventional CFD technology has enabled precise numerical analysis of attached flows at relatively low angles of attack. In 2000, there was a CFD workshop held at National Aerospace Laboratory in Japan (website in Japanese, http://www.nal.go.jp/cfd/jpn/CFDWeb/2dstall/cntnframe.htm). In this workshop, flows at low to high angles of attack were the targets to simulate for three types of wing sections having different stall characteristics; NACA63-018, NACA63-012, and NACA64A006. The stall characteristics of the NACA63-018 and the NACA63-012 are classified into trailing-edge stall and leading-edge stall respectively, and were well predicted by the conventional Reynolds-Averaged Navier-Stokes (RANS) computations. However, prediction of stall characteristic of the NACA64A006 airfoil at high Reynolds numbers was not successful. This airfoil has a thin-airfoil stall feature, where laminar flow separation occurs at the leading edge and transition makes turbulent reattachment. The reattachment point gradually moves rearward with angle-of-attack increase.

Figure 4 shows the summary of the lift characteristics of NACA64A006, thin airfoil computed by many RANS models at the workshop. The wing configuration is simple and grid generation is not a difficult task. However, simulations are not easy. We will come back to this point in Section 4.

3.3.2 High-α Flows over a Delta and Double-delta Wings

The characteristics of a delta wing at low speed and relatively high angles of attack are governed by leading-edge separation vortices (Fig. 5). A lot of experiments and numerical simulations have been conducted and the characteristics of the flows over a delta wing have been discussed. Fairly accurate results were obtained for a simple delta wing[7,8], where the lift and moment characteristics were well predicted by the Navier-Stokes simulations as they capture the growth of leading-edge separation vortices that are the key mechanism of non-linear lift production. However, spanwise pressure distributions over the upper surface of the wing depend on the grid resolution or computational schemes. Figure 6 shows one example of the simulation results for the delta wing with aspect ratio 1 (76 degrees sweep- so called Hummel's case). There are many reasons for the discrepancy between the computational results and the experiment, but grid resolution obviously influences to the vortical-flow structures over the upper surface of the wing. Fortunately, lift and moment characteristics are well predicted even by the simulations using coarse grid distributions. Figures 7(a) and 7(b) are the computed locations and the strengths of the leading-edge separation vortices at each chordwise station. The result shows that vortex is weaker for the simulation using coarse grid but the vortex core is located closer to the wing surface, resulting in the similar lift force due to the vortex. This was pointed out by the present author in 1987 for the double-delta wing[9]. Although lift and moment characteristics of a simple delta-type wing can be well predicted by the Navier-Stokes simulations, it does not necessarily mean that the flow field is well captured. It is still mystery to the author if the agreement of the lift is accidental or there may be a physical reason (such as conservation of vorticities).

The situation becomes worse for a double-delta wing having a kink in the leading edge. Over the double-delta wing, there exist two vortices emanating from the leading edge of the strake and the main wing (see Fig. 8). These vortices interact each other and finally merge together at certain angles of attack. Thus, the flow field over a double-delta wing is more complex. Numerical simulations to study the flow field in the past showed there still remains large discrepancy with the experiment even with the simulation using very fine grid distributions[10].

We see a lot of nice visualization images of leading-edge vortices for fighter-type aircraft, but we should remember that even the flow field over a simple double-delta wing is not necessarily well simulated. The aerodynamic forces and moments may show good agreement, but the flow phenomenon may not be well captured in the simulation. The base of such simulation is still rather fragile compared to the attached flows over a conventional wing.

3.3.3 Supersonic Base Flows –Finding a New Efficient Tools-

Base regions exist in most of the space transportation vehicles. Accurate simulation of the base flows is critically important as the base drag influences to the aerodynamic characteristics of the vehicle at certain speed range. As shown in Fig. 9 schematically, supersonic base flow includes a large recirculation region. As the freestream is supersonic, the interaction of shear layer with expansion and compression waves appear. Even with such practical importance, estimation of the base pressure has not been successful until recently. We have carried out the simulations for HOPE-X experimental vehicle, aerospike nozzles and other flows including the base region using Raynolds-averaged Navier-Stokes (RANS) model. However, the pressure distributions over the base area were not well captured in any simulation. Similar report was published in Europe.

3.3.4 Observations from the examples

From the three examples shown above, we notice that there still remain problems, which are geometrically simple but difficult to simulate by CFD. We can easily obtain the numerical solutions, but the reliability stands on a fragile base. Even with less accurate solutions, we may find important physics or we may find some data useful for design and analysis, as reliability really depends on the people to use CFD. However, insufficient solutions (like improper model or insufficient grid resolution) are sometimes used for the discussions that should not be done based on such simulations. Validation of the physical and mathematical models and verification of the numerical solutions are discussed in many CFD workshops. They have shown some guidelines for specific applications especially from the viewpoint of computational mechanics. Most of them are not discussed from real engineering viewpoint. Unfortunately, the threshold has not been established in many flow physics, and the decision depends on researchers' experience. We always have to foresee the insufficiency behind CFD simulations.

4 Paradigm Change for the Future Prospect

4.1 Evolutional Effort RANS to LES/RANS Hybrid

There is an obvious shift in the CFD research from Reynolds-averaged Navier-Stokes Simulations (with turbulence model) to Large Eddy Simulations with the computer progress. As Phillip Spalart suggested in Ref. 11, emerging technology is hybrid strategies retaining conventional turbulence modeling in attached region while adopting a large eddy simulations in the region of separated flows. The examples in the section 3.2 all includes large recirculating flow regions and are good test cases for such hybrid strategies. The shift is supported by the rapid progress of computer performance, but more importantly, we start to recognize that flow physics, even from the engineering viewpoint, requires unsteady flow simulations. Separated flows are inherently unsteady and steady recirculating region may be the result of an average of strongly unsteady flows. Capturing such unsteady flow behavior is inevitable for the flow analysis and eventual control of the flows. Without LES that captures unsteady behavior of the flow, accurate result may not be obtained even as an average as the following examples will show.

With the rapid progress of computer capability in recent years, LES has been applied to the flows of some airfoils near stall at high Reynolds numbers[12-14]. Reference 12 suggested that LES can successfully resolve the turbulent transition directly and predict the flow behavior including separation and reattachment, if the mesh is adequately fine near the walls to resolve near-wall turbulent structures while the results with coarse mesh resolution are generally disappointing. However, the mesh resolution required for LES becomes enormous (from our estimation of this problem, it would require about 500 times more grid points necessary than the computation shown here). In addition, fine mesh resolution near wall limits the time step size. Therefore, it remains difficult to apply LES to complex flows at high Reynolds numbers as seen in many engineering problems under the current computer environment. To overcome these difficulties, LES/RANS hybrid methodology was proposed in recent years. Present hybrid method is different from Detached-Eddy Simulations which is the simple modification of Spalart-Allmaras turbulence model[15]. The hybrid approach is a relatively new method in which the RANS formulation is applied near the solid surface, while the LES formulation is applied to massively-separated flow regions. The hybrid methodology is considered to need less computational cost than LES as it alleviate the required mesh resolution near wall and the resultant time step limitation. LES/RANS hybrid methodology was successfully applied to supersonic turbulent mixing flow[16].

4.1.1 Thin-airfoil Stall Characteristics – RANS/LES hybrid Simulations

As presented in the section 3.2, estimation of thin-airfoil stall characteristics using any RANS model has not been successful. Here, LES/RANS hybrid methodology is applied to subsonic flows over a NACA64A006 airfoil at high Reynolds number and various angles of attack. As pointed out earlier, it is important to capture laminar flow separation, transition and turbulent reattachment near the leading edge. To improve the spatial resolution, spacial derivatives of governing equations are evaluated by the sixth-order compact difference scheme, which achieves the spectral-like resolution with minimize dispersive and dissipative numerical errors, proposed by Lele [17]. The computational conditions is set to same as the experiments of McCollough and Gault.[18,19]. Figure 10(a) shows the instantaneous plots of the isosurface of the total pressure in space and the pressure contours over the wing surface. There observed are small vortices emanating from the location near the leading edge and constructing a sheet-like structure. When the flow is averaged for a certain time span, the flow variation in the spanwise direction disappears and the flow becomes almost two-dimensional as Fig. 10(b) shows. In this “averaged” flow, there exist a small bubble near the leading edge as is observed in the chordwise Mach-number contours plotted in Fig. 11. Finally, the computed CL-α curves are compared with the experiment in Fig. 11. The RANS/Hybrid simulation successfully captured the thin-airfoil characteristics. The results computed with the conventional upwind scheme also shown in Fig. 12 fails to predict lift characteristics, and the result indicates that simulations using conventional upwind method would require much higher grid resolution.

The much more detailed discussion of the flow structures as well as the computational method is presented in the original paper[20].

4.1.2 High-α Flows over a Delta and Double-delta Wings –not yet Solved

The author did some work in late 80’s[21,22] but has been away from the simulations of high-α delta wing flows since then. However, the poor results (in the sense of flow physics) in late 80’s stayed in his mind and therefore simulation efforts started again a few years ago. Firstly, we improved the grid resolution as the former effort used less than one million grid points. As presented in Ref. 10, four million background grid points with another four million grid points locally adapted to the vortical flow region were used for the simulation. The result obviously showed some improvement but was not satisfactory. Then, we applied compact difference scheme (as the simulation of thin-airfoil characteristics shown above) to reduce the computational cost[23].

The results for a simple delta wing and a double-delta wing are shown in Figs. 13(a) and 13(b) as spanwise surface pressure distributions at certain chordwise location over the wing surface. Both results show another improvement, but the location of the main vortex still showed different from the experiment. The computed locus of the strake and wing vortices and resultantly spanwise pressure distributions in another chordwise stations showed difference. Note that we have been used RANS model and/or laminar flow simulations so far. With improvements in the measurement techniques, there appeared good experimental data available. For a delta wing, Mitchell et all[24] showed the presence of stationary sub-structures in the rolling-up shear layer as a leading-edge separation vortex in mean flow measurement at near one million Reynolds numbers. They also did computations using DES (Detached Eddy Simulation; a kind of LES/RANS hybrid methods) and showed both steady and unsteady sub-structures exist in the shear layer depending on the grid resolution[25]. Visbal and Gordinier computed the flow at a low Reynolds number using compact difference scheme[26] and studied the vortex structure in the shear layer. As these results suggested and the author pointed out in the beginning of this section, we may need to resolve the structure of the rolling-up shear layer for accurate prediction of the leading-edge separation vortex flows over a delta wing. The approach should use LES or RANS/LES hybrid method and such effort is underway. The leading-edge separation vortex flow structure is much more complicated than we expected and accurate simulation is very difficult even for a simple body configuration.

4.1.3 Supersonic Base Flows -RANS/LES hybrid Simulations

As pointed out in section 3.2, base pressures of a blunt body at supersonic speeds has not been predicted well. There are various techniques for the numerical prediction of such complicated turbulent flows but RANS/LES hybrid method may be a good choice.

The experiment[27] by Herrin and Dutton for the axisymmetric base flow is taken here as an example for the simulation. Free-stream Mach number of M=2.46 and a unit Reynolds number of 45 million per meter imposed as inflow. With the base radius of 31.75mm, the resulting Reynolds number based on the diameter is 2.858 million. The computational grid used in this study is roughly three million[28].

Figure 14 shows an instantaneous view of the computed vorticity magnitude contours computed by the LES/RANS hybrid method. There exists strong flow unsteadiness and the flow field changes in time although only one shot is shown here. Figure 15 shows the same plot computed by the RANS method. The computed flow field is steady and does not change in time. When a sequence of the time-dependent results computed by the LES/RANS hybrid method for certain time span is averaged, the vorticity magnitude contours become similar to Fig. 14 (although the size of the recirculating region is different as will be discussed later). This indicates that there exist strong unsteadiness in the supersonic base flow and the steady flow field that we usually observe and is known as a typical supersonic baseflow is a time-averaged flow field. Note that the visualized image of the crossflow in the wake created from the computed result shows strong unsteadiness and it showed good agreement with the image taken in the experiment. The time-averaged base pressure distributions along the base surface are compared with the experiment in Fig.16. The RANS computation shows the lower pressure distributions compared to the experiment. Also there is a variation of the pressure distribution over the base surface. Although not shown here, the RANS results shows that strong reverse flow exists in the wake region. The flow is accelerated outward along the base surface, which causes the reduction and variation of the pressure in the radial direction as in Fig. 16. Note that "LES" in this plot is the solution using LES with insufficient grid resolution near the wall. The LES/RANS hybrid method shows very good agreement with the experiment.

4.1.4 Observations from the examples

The results showed that capturing unsteady nature of the flow field leads to accurate flow simulations and LES/RANS hybrid method including DES method is appropriate for accurate simulation of complex flows under the reasonable computer resources and computer time. The LES/RANS hybrid method may replace many of the RANS simulations and can be a practical simulation tool in the near future.

4.2 New Considerations

4.2.1 further than Wing Design - CFD database as an example

Our Institute has a research effort to develop reusable launch vehicles (RLV). Although nothing has been approved officially as a main project, we continue efforts toward the real launch. We believe CFD will play an important role in the design process as changing the flow conditions or body geometries are rather easy in CFD. In addition, CFD would tell us more about the flow fields and help our understanding for the aerodynamic characteristics.

In the past, CFD has not been used as a key design tool of rockets and/or spacecraft in our Institute although it has been used to supply additional data to the experiment. CFD has been mainly used to confirm that the design methodology is acceptable, or used for the analysis of element by element. One of the main reasons for CFD not being involved in the design process is that accuracy of CFD simulations was not well established. Accuracy of CFD simulations over aircraft was discussed frequently and, for example, it was shown that the estimation error of transonic drag of wings by the Navier-Stokes simulations is less than certain counts[29]. For the body configurations used as space transportation systems, we cannot clearly tell how accurate we can predict the forces and moments even for a very simple configuration. The people in design are not interested in sophisticated turbulence models, but they need to know a clear answer if the estimation error is 5%, 20% or 100%. They would accept conventional simple turbulence model if the solution accuracy is clearly given. If the error were to be, for instance, within 10%, CFD simulations would be used for the initial estimation of the flight path design of reusable launch vehicles (RLV).

Figure 17 shows an example of SSTO (Single stage to orbit) body configurations. Even with such simple configurations, we cannot clearly tell accuracy of the CFD simulations. It is important to discuss effect of nose radius, shoulder curvature, base flatness and other geometrical parameters since the pitch angels would vary 360 degrees in its maneuver. There are two key issues here. First, we need to accumulate a lot of computational data for wide variety of body geometries and flow conditions. Second, certain level of reliability should be established based on the discussion on the accuracy of the data. As the body configuration is simple, simulations are not difficult. Neither sophisticated grid generation program, nor sophisticated computer code is needed. From the viewpoint of CFD technique, it may not be interesting. However, the results of such "easy" simulations will be useful in the real design of RLV's. We recently conducted many CFD simulations in that direction for current and future space transportation vehicle development.

Since the purpose of this paragraph is not to show the detailed results but to show the importance of this kind of work as a new direction of CFD research, very limited data is presented. We first simulate the flow field over an Apollo capsule. Although Apollo capsule is not designed for a reusable vehicle, it was chosen as a representative configuration of bluff bodeis and there is large amount of experimental data available for comparison. The computed axial force coefficients for the capsule-like configuration are shown in Fig. 18 for validation purpose. The CFD results are in excellent agreement with the experimental data[30,31] for all angles of attack range at whole speed regime. The results for wide variety of Mach numbers and angles of attack indicated that the CFD aerodynamic prediction at least for the basic configuration has sufficient reliability for the preliminary design of RVT's. Although not shown here, some of the configuration parameters are changed for the discussion and the detailed flow mechanisms behind the characteristics are investigated[31]. A lot of computed data for many body configurations at wide range of flow conditions is accumulated and CFD database is under construction for the future design.

The author emphasized the shift from RANS to LES-type simulations in this paper, and it may sound contradictory to carry out RANS simulations using coarse grid points. Accuracy of the CFD study is frequently discussed based on the comparison with the experiments from computation viewpoint. The author believes the accuracy depends on the requirement from engineering viewpoint. For the preliminary design of RVT's, the most important issue is that we need to supply a tool that we guarantee its accuracy. Simulations tools that accurately capture flow physics are to be used in the further stages of the design process. We need to develop CFD technology toward two directions; pursuit for better accuracy for the detailed design and physical discussions and confirmation of the accuracy of conventional approach as an current engineering tool. Sometimes, they are mixed up in the discussion on CFD.

4.2.2 Tools to Help Design

For the design of commercial aircraft, the author believes that basic body configuration is not too much different from existing ones. Requirement of CFD in the design of future aircraft may be to improve the aerodynamic configuration using CFD as an analysis tool fulfilling the new design requirements. There are a lot of data available and the designers can start with such data. The design may be conceptually same as the existing ones, and optimization methods using CFD may work fine.

Suppose we develop an aircraft with flapping wing, what should we do? Suppose we develop a aircraft on Mars, what should we do? There is little accumulation of the former experimental and computational data available. We probably find a book and see the basic theory to get a good idea. It would require long time. CFD or EFD can be used after such idea is determined. From the discussion with non-CFD aerodynamic researchers, the author found that there is a strong requirement to use CFD as a tool for the conceptual design. They need a tool for help their thinking. They know flow physics but do not want to know CFD details. To make CFD as a such tool, we have to develop an infrastructure or a system that does not require the knowledge of CFD details such as grid distributions, turbulence models, and so on. Such system depends on the applications and therefore commercial software is not adequate as such software aim to handle any problems. An example may be shown in the presentation[33].

4.2.3 Education of Fluid Dynamics

With the same reason, CFD can be used as a good teaching tool for fluid dynamics. Such was done in universities but should be done in industries as well. As the CFD solutions are based on the physical and mathematical models, and include discretization errors. Therefore, the use should be carefully done. With such cares, CFD would become a very good education tools both for students and engineers. Again, we need to develop the infrastructure or software system to realize it.

4.3 Revolution?

Honestly speaking, we have not clearly found what kind of revolution may occur and change the trend of CFD research. There may be some clues.

We, CFD researchers should remember that CFD is only a small part of the Aircraft/Spacecraft design even though aerodynamics plays an important role in aerospace sciences. There is a strong trend to combine CFD method with other physical disciplines; fluid-structure coupling, fluid-combustion coupling, fluid-flight dynamics coupling and else. Such interdisciplinary analysis is really important for practical problems. Same is true for Optimizations. We see a lot of such research examples, but both interdisciplinary and optimization researches are applied to selected topics. When considering design and development process of conceptually new product, we first need to analyze the R & D process and find out where sophisticated CFD is necessary and where low-order analysis is sufficient. CFD researchers should spend more time for understanding the R & D process. Optimization becomes really useful when being combined with Reliability-based Design which requires another computational mechanics analysis.

Since bottleneck of the time required for the flow simulation the period required for preparing the surface and volume grid data from the CAD geometry, revolution may occur there. Currently, many people try to use Cartesian-type grid to avoid lengthy and tough body-fitted grid generation. The difficulty exists in capturing proper viscous layers over the body surface, but if this were solved, that will totally change to CFD simulations in aerospace.

5 Conclusions

Historical perspective of CFD in aerospace in last 30 years was given. CFD in aerospace has lead CFD in whole the engineering in the past. In 1980's and 90's, a lot of nice ideas came out from the CFD researchers in aerospace and spread into other engineering fields. Now, it seems that CFD in aerospace does not seem to have such leaderships in CFD community.

Dean Chapman, one of the pioneers and founders of CFD in aerospace wrote, in his "Opening Remarks" in the workshop late in 70's that there are two major motivations behind CFD and it would not change in coming decades. They were (1) providing an important new technology capability and (2) economics. The issue is true and many have been realized. However, CFD has not shown its superiority well to wind tunnel experiments.

Even though there appear a lot of simulations for complex body configurations in these days, there still remain quite a few simple problems that are difficult to simulate. There is a strong shift from RANS simulations to LES and/or RANS/Hybrid methods due to the progress of computer performance and that shift may solve some of the problems under the current or near-future computer environment. This is happening not only because of the progress of computers but also because of our recognition that separated flows are inherently unsteady and successful simulations require LES-like computations.

CFD researchers have been trying to prove the capability of CFD showing examples for complex body configurations. However, there may be other types of research necessary for CFD to become a real useful tool for the design. As one of the examples, construction of CFD database was presented. Another issue was to make a CFD infrastructure so that people outside CFD community may use CFD as a tool to hit or refine their ideas.

We have not found the clear clue for the revolution of CFD research but that may come out from the requirement by the design and development process.

References

[1] Lynch F. T. Chapter II " Chapter II Commercial Transports – Aerodynamic Design for Cruise Performance Efficiency", *Transonic Aerodynamics*, Ed. by D. Nixon, Progress in Astro. & Aero., Vol. 81, AIAA, 1982 .

[2] Chapman D., 'Opening Remarks' Future Computer Requirements for Computational Aerodynamics, NASA CP 2032, 1978.

[3] Aerospace America, "The year in Review: Fluid Dynamics and Applied Aerodynamics, December 1986.

[4] Fujii K. and Obayashi S., "Navier-Stokes Simulations of Transonic Flows over a Practical Wing Configuration", AIAA Journal, Vol. 25, No.3, pp. 369-370, March, 1987.

[5] Fujii K. and Obayashi S., "Navier-Stokes Simulations of Transonic Flows over a Wing Fuselage Combination", AIAA Journal, Vol. 25, No.12, pp. 1587-1596, December, 1987.

[6] Murman, E. M., ''Challenges in the Better, Cheaper Era of Aeronautical Design Engineering and Manufacturing', CD-ROM proceedings of the 22nd International Congress of Aeronautical Sciences, Harrogate, U.K., August 2000.

[7] Thomas, L, Taylor, S. L. and Anderson, W. K.,"Navier-Stokes Computations of Vortical Flows over Low Aspect Ratio Wings", AIAA Paper 87-0207, 1987.

[8] J. A. Ekaterinaris J. A. and Schiff L. B., "Numerical Prediction of Vortical Flow over Slender Delta Wings", Journal of Aircraft, Vol.30, No.6, Nov.-Dec. 1993.

[9] Fujii, K. Gavali, S. and Holst, T. L., 'Evaluation of Navier-Stokes and Euler Solutions for Leading Edge Separation Vortices', International Journal of Numerical Methods in Fluids, Vol. 8, No. 10, pp. 1319-1329, October, 1988.

[10] Horie, T. Fujii, K. and Hattori, N., 'Numerical Simulations of Leading-Edge Separation Vortices' JSASS 15th International Sessions in 39th Aircraft Symposium Incorporating 2nd Japan-Korea Aerospace Technology Symposium , October, 30, 2001.

[11] Spalart P., 'Strategies for turbulence modeling and simulations', Int. J. Heat Fluid Flow, 21 (in press), 2000.

[12] Mellen P. C., Frohlich J. and Rodi W., "Lessons from LESFOIL Project on Large-Eddy Simulation of Flow Around an Airfoil" AIAA Journal, Vol. 41, No .4, April 2003, pp.573-581.

[13] Mary I. and Sagaut P., "Large Eddy Simulation of Flow Around an Airfoil Near Stall", AIAA Journal, Vol. 40, No. 6, June 2002, pp.1139-1145.

[14] Dahlstron S. and Davidson L.,"Large Eddy Simulation of the Flow Around an Airfoil", AIAA Paper 2001-0425, Jan. 2001.

[15] Spalart P. R. and Allmaras S. R.,"A One-Equation Turbulence Model for Aerodynamic Flows", AIAA Paper 92-0439, Jan. 1992.

[16] Georgiadis N. J, Alexander J. I. D. and Reshotko E.,"Hybrid Reynolds-Averaged Navier-Stokes/Large-Eddy Simulations of Supersonic Turbulent Mixing", AIAA Journal, Vol. 41, No. 2, Feb. 2003, pp.218-229.

[17] Lele K. S.,"Compact Finite Difference Schemes with Spectral-like Resolution"`, Journal of Computational Physics, 103, 1992, pp. 16-42.

[18] McCollough G. B. and Gault D. E., "Examples of Three Representative Types of Airfoil-Section Stall at Low Speed"', NACA TN2502, 1951.

[19] McCollough G. B. and Gaul D. E.,"Boundary-Layer and Stalling Characteristics of the NACA64A006 Airfoil Section"', NACA TN1923, 1949.

[20] Kawai, S. and Fujii K., " Stall Prediction of a thin Airfoil Using LES/RANS Hybrid Methodology with Compact Difference Scheme", AIAA Paper 2004-2714, June 2004.

[21] Fujii K. and Kutler P., Numerical Simulation of Leading-Edge the Separation Vortex for a Wing and Strake-Wing Configuration AIAA 6th Computational Fluid Dynamics Conference, Danvers, MA, U.S.A., AIAA Paper 83-1908-CP, July 1983.

[22] Fujii K. and Schiff L. B., Numerical Simulations of Vortical Flows over a Strake-Delta Wing AIAA Journal, Vol. 27, No. 9, pp. 1153-1162, September, 1989.

[23] Arasawa T., Fujii K. and Miyaji K., "Application of High-Order Compact Difference Scheme to Vortical Flow Computation over a Delta and Double-Delta Wings", AIAA Paper2003-3537, June, 2003 (to appeaer as an article in Journal of Aircraft).

[24] Mitchell A., Molton P., Barberis D. and Delery D., "Characterization of Vortex Breakdown by Flow Field and Surface Measurement", AIAA Paper 2000-0788, 2000.

[25] Mitchell A., Morton S. and Forsythe J., "Analysis of Delta Wing Vortical Substructures Using Detached-Eddy Simulation", AIAA Paper 2002-2968, 2002.

[26] Visbal M. R. and Gordnier G. E., "On the Structure of the Shear-Layer Emanating from a Swept Leading Edge at Angle of Attack", AIAA Paper 2003-4016, 2003.

[27] Herrin, J. L. and Dutton, J. C., "Supersonic Base FlowExperiments in the Near-Wake of a Cylindrical Afterbody", AIAA Journal, Vol. 32, No. 1, Jan. 1994.

[28] Kawai S. and Fujii K., "Computational Study of a Supersonic Base Flow Using LES/RANS Hybrid Methodology", AIAA Paper2004-0068, Jan, 2004.

[29] Warfield M. J., "The year in Review, Applied Aerodynamics", Aerospace America, pp. 12-13, December 2001.

[30] Fujimoto K, Fujii K. and Tsuboi N.,"CFD Prediction of the Aerodynamic Characteristics of Capsule-like Configurations for the Future SSTO Developments", AIAA Paper 03-0912, 2003.

[31] Fujimoto K and Fujii K.," Compressible Flow Simulations Over Basic Reusable Rocket Configurations", JFEC03/DAC-1234, 4th ASME/JSME Joint Fluids Engineering Conference, July 2003.

[32] Fujii K and Miyaji K.,"WEB-CFD and Beyond-CFD for non –CFD Researchers", AIAA paper, 2002-0753, 2002.

Table 1 Wind tunnel experiment and Computational fluid dynamics

EFD	**CFD**
Wind tunnels	Computers
Measurement techniques	Numerical algorithms
Manufacturing techniques	Programming techniques (parallel language…)
Model manufacturing	CAD interface, Grid generation
Data acquisition	Post-processing
Data handling	Visualization software
Reynolds number effect	Discretization error, turbulence model, …

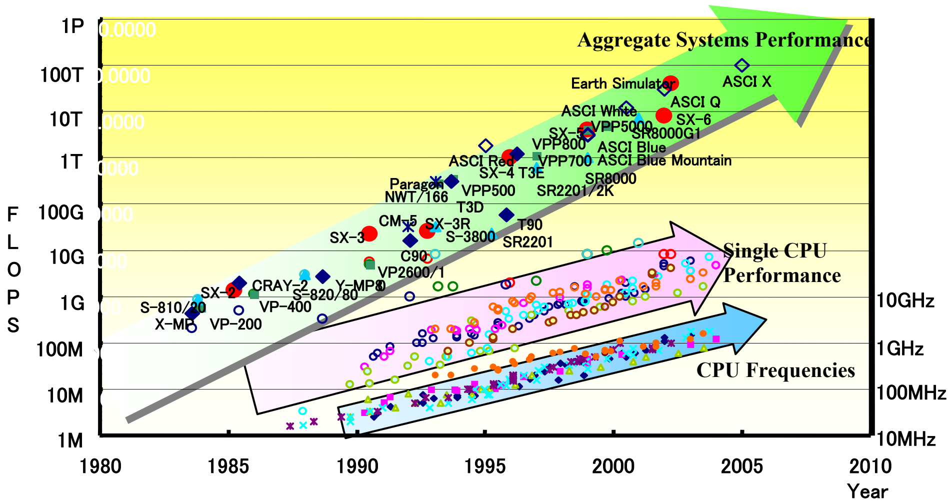
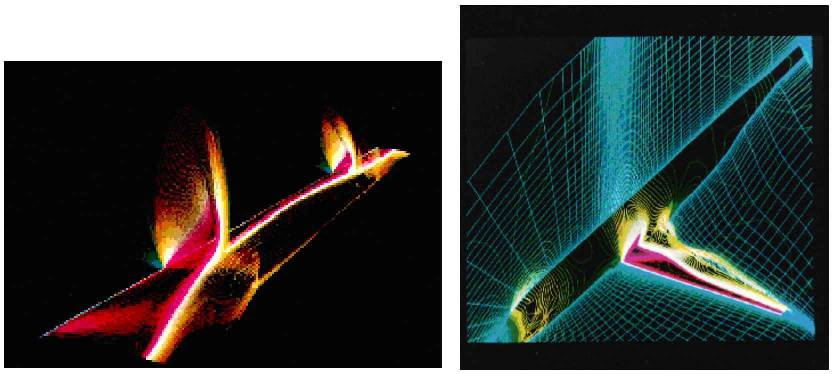


Figure 1 Progress of Computers



(a) practical wing (b) practical wing-fuselage

Fig. 2 Three-dimensional Navier-Stokes simulations in 1986

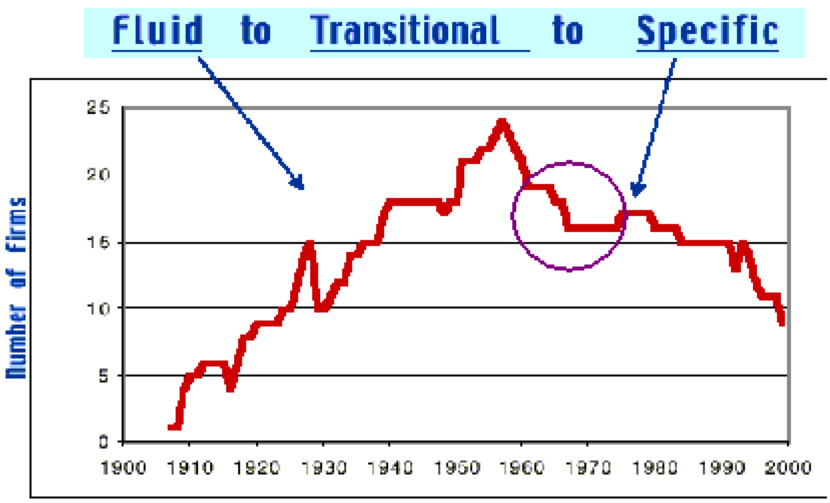


Fig. 3 Evolution of CFD - Utterback's theory[6]

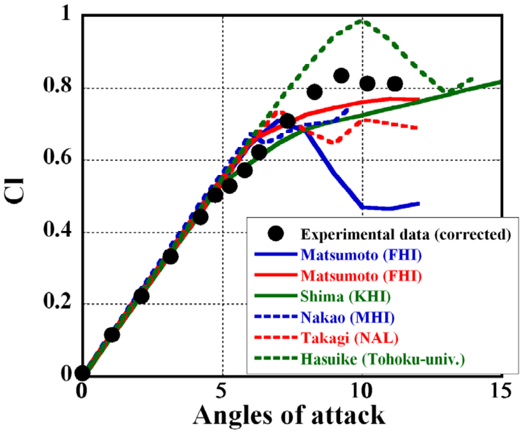


Fig. 4 Estimation of the lift characteristics by RANS simulations: taken from the web site

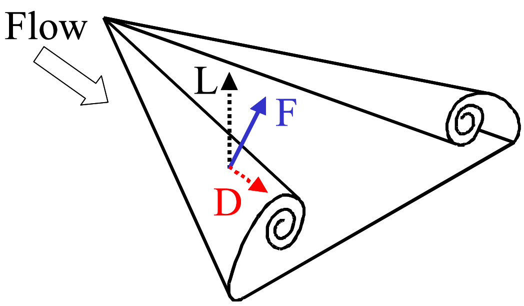


Fig. 5 Schematic picture of vortical flow field over a delta wing

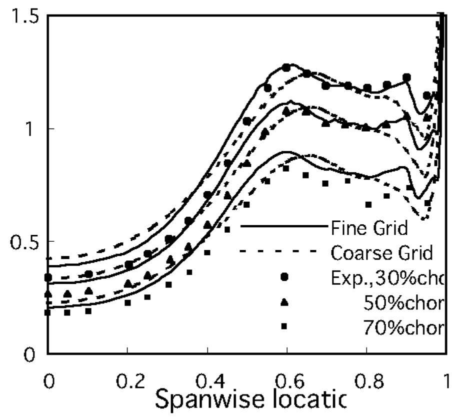
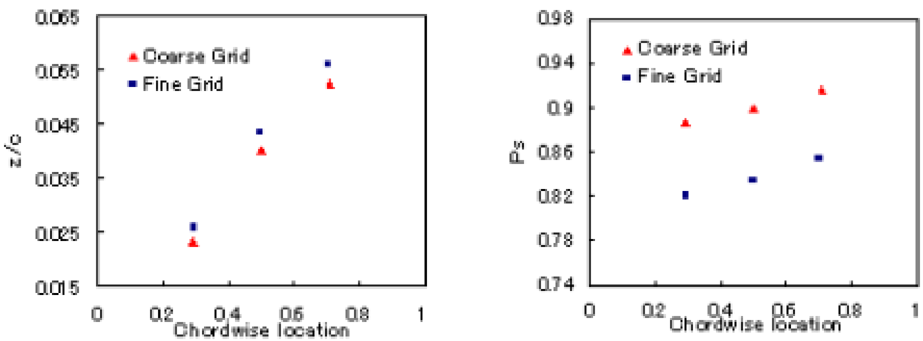


Fig. 6 An example of the simulation result –spanwise pressure distributions



(a) Location of vortex cores (b) Strength of vortex

Fig. 7 Effect of grid resolutions

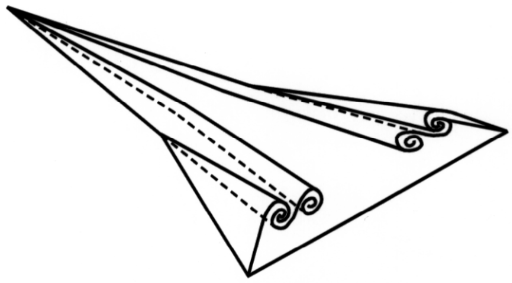


Fig. 8 Schematic picture of vortical flow field over a double- delta wing

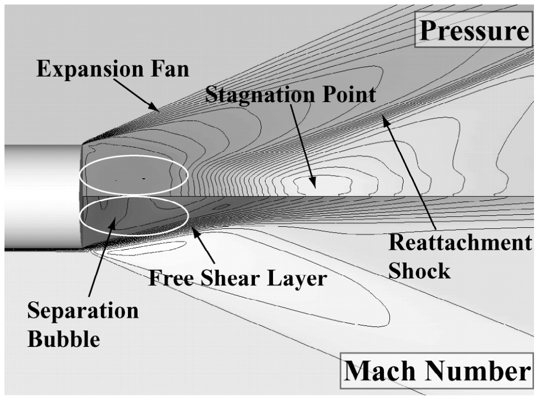
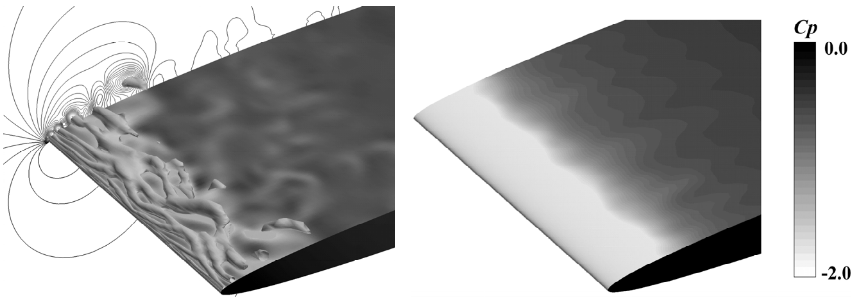
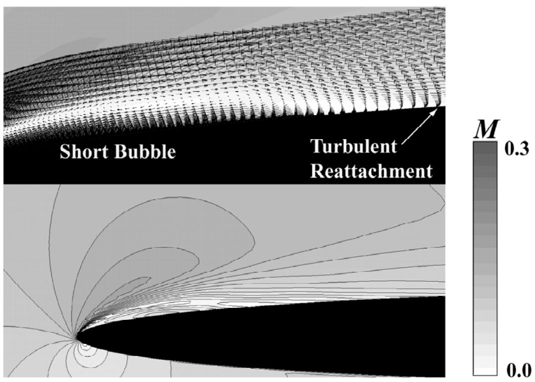


Fig. 9 Schematic picture of supersonic base flows



(a) instantaneous (b) time-averaged

Figure 10 Simulation result by the LES/RANS hybrid method with compact differenceing



chordwise plots of the time-averaged solution

Figure 11 Simulation result by the LES/RANS hybrid method with compact differenceing

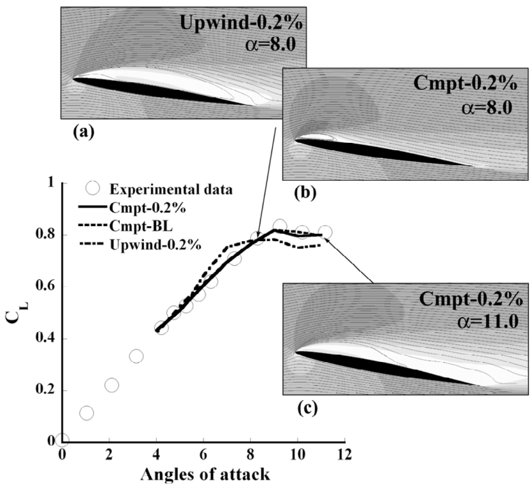
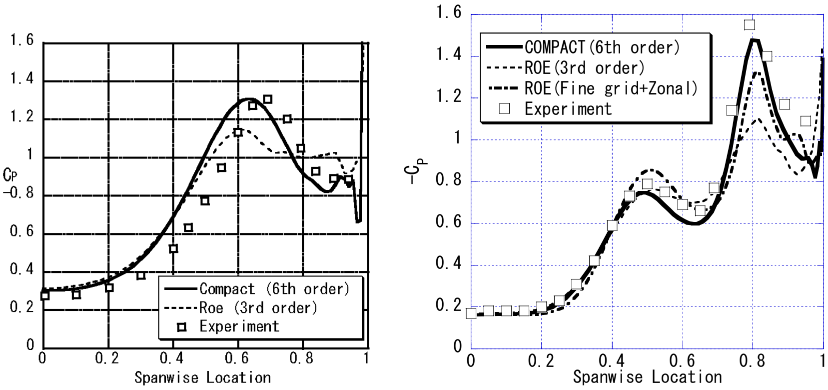


Figure 12 CL-alpha curve by the LES/RANS hybrid method with compact differenceing



(a) delta wing (b) double-delta wing

Fig. 13 Spanwise CP distributions

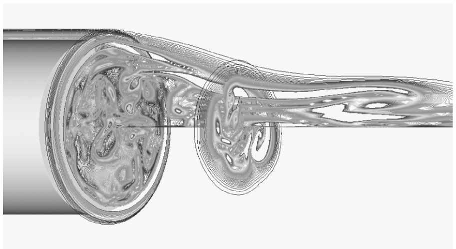


Fig. 14 Instantaneous view of the vorticity magnitude contours: LES/RANS hybrid computation

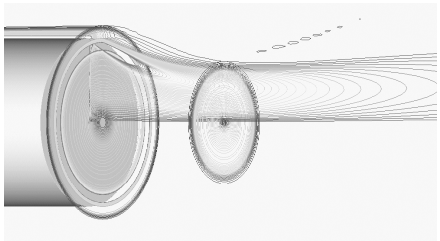


Fig. 15 Instantaneous view of the vorticity magnitude contours: RANS hybrid computation

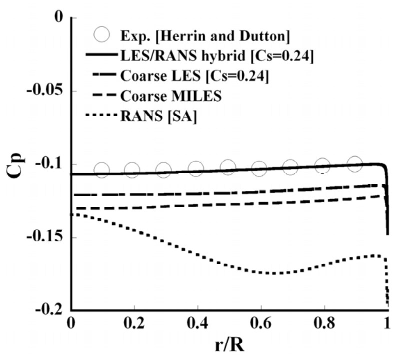


Fig. 16 Comparison of the computed base pressure distributions

Fig. 17 SSTO Reusable Launch Vehicle

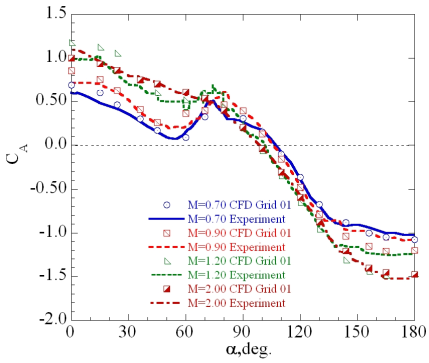


Fig. 18 Computed axial forces for the Apollo capsule