CFD contributions to high-speed shock-related problems
- examples today and new features tomorrow -

K. Fujii

Institute of Space and Astronautical Science
Japan Aerospace Exploration Agency
Sagamihara, Kanagawa 229-8510, JAPAN

Summary. Computational Fluid Dynamics (CFD) have extensively contributed to high-speed shock-wave research. With study examples by the author's group in the past, effectiveness of CFD both for design of transportation vehicles and for understanding of fluid physics is discussed. Trend of CFD for further use is then discussed based on recent applications and three key features: computer progress, spectral-like high-resolution scheme and LES and LES/RANS hybrid method are focused. Recent CFD research reveals that high speed flows even the ones considered to be steady state have inherently unsteady nature that requires LES-like computations for successful simulations. Such simulations require remarkably higher grid resolution but emerging numerical techniques having spectral-like high-resolution would help reducing the number of grids required for such simulations. The paper is summarized by the address to the issues about future CFD.

1 Introduction

Computational Fluid Dynamics (CFD) study for high-speed flows in aerospace engineering was initiated in early 70's. The embedded shock wave was automatically captured in the computer simulation [1] and the design process of commercial aircraft was drastically changed. CFD study for shock-related problems was initiated much earlier before that . For example, Peter Lax developed a technique for computing high-speed flows including shock waves which represent discontinuities in the flow variables [2] in 1954. At any rate, a lot of CFD studies appeared especially after supercomputers was developed in late 1970's. Some are CFD study for fundamental shock-related problems and some are practical applications on shock-related problems. Theoretical and experimental studies had been two main pillars of research tools and CFD became a third pillar with the progress of computers. Now, CFD simulations became much easier than before [3] and even a small academic group can make important contributions to shock wave study.

Conventional RANS(Reynolds-averaged Navier-Stokes) simulations using turbulence models are widely spread as a useful tool for many research areas. With this as a background, CFD technology in high-speed flow research is considered to be in the first matured stage. Former paper by the present author looked back the CFD history and propose a new direction of CFD in the future [4]. When considering difference between CFD studies in 1980's and 2000's, we surprisingly notice that there is a little difference except that CFD simulations became much easier for more complex geometries. However, with three emerging technologies: computer progress, spectral-like high-resolution schemes and LES and LES/RANS hybrid methods, CFD for high-speed flow problems has started to move to a new stage. In the following sections, some of the author's studies are presented to show how CFD studies would shift its direction under the trend above.

2 Example in the Past - CFD became a good analysis tool

Present author has been engaged in the CFD study for high-speed trains for many years [5]- [7]. Some of the earlier results were used for the design of front cars of Shinkansen (rapid) trains in Japan. Recently, magnetic levitation trains are under development by Japan Railway Central (JRC) and extensive CFD study for the aerodynamic problems are conducted under the collaboration between JRC and the author's laboratory in ISAS/JAXA (Fig. 2).

Initial effort in 1990's was a CFD studyfor a aerodynamic problem of the train entering a tunnel. Figure 1 shows the schematic picture of the flow mechanism [5]. When a train enters a tunnel at high speed, strong pressure wave is created in front and it propagates toward the exit of the tunnel. The pressure wave tends to become steeper and strong booming noise appears outside the exit. The booming noise is one of the important aerodynamic problems for the development of Shinkansen trains as there are many tunnels in Japan. Comparison with the existing theories, experiments and field measurements showed that the CFD approach was useful for understanding the mechanism and for finding a method to alleviate the formation of compression wave inside a tunnel. Based on the simulation results, a new theory called v_{wall} theory was developed.

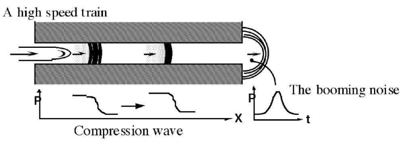

Fig. 1. Mechanism of the booming noise generation

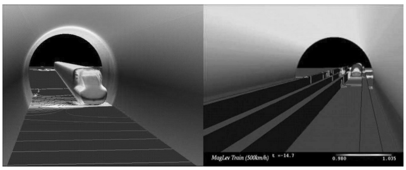

Fig. 2. Example of the simulation for Shinkansen and Maglev trains

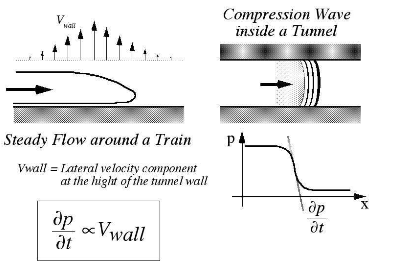

Fig. 3. Pressure gradient and the v_{wall}

Only the essence of the v_{wall} theory is presented [5], [7]. Let's consider change of instantaneous streamlines in time. When a tunnel comes closer to the train, it shrinks the streamtube near the train. Pressure increase inside the tunnel occurs due to this streamtube being narrowed. There exists outward velocity component near the train running outside the tunnel and that is diminished after the train enters the tunnel. Here, we define the integral of the outward velocity component normal to the tunnel wall as v_{wall} as schematically shown in Fig. 3. Some mathematical manipulations show that the gradient of the compression wave inside the tunnel is proportional to the v_{wall}. We call this to be "v_{wall} theory".

In summary, this theory can predict formation of the compression wave accurately only based on the steady-state flow filed. This newly-developed v_{wall} theory became a good engineering tool for the front car design alleviating the booming noise of high-speed trains. CFD gives us large three-dimensional data from which widen our views on flow features and help our understanding of fluid physics in addition to its capability as an analysis tool.

3 Evolutionary effort and New trend in CFD

As shown in the previous section, conventional RANS(Reynolds-averaged Navier-Stokes) simulations are widely spread and CFD in high-speed flow research is considered to be in the first matured stage. Gradual change that moves CFD to on the second matured stage occurs recent years. There are three key features behind it: computer progress, spectral-like high-resolution scheme and LES and/or LES/RANS hybrid method.

First factor: computer progress

In this section, a study example that became possible with the first key feature "computer progress" is presented. Simulations for problems including complex shock waves required large amount of computer time and therefore the discussions have been based on the restricted number of cases. With the progress of computer hardware and related software, studies with number of cases with different flow and/or geometrical parameters became feasible.

The authors' group has been engaged in both experimental and numerical studies of under-expanded jets impinging on an inclined flat plate [8], [9]. Flow fields of the M2.2 jet impinging on an inclined flat plate at various plate angles, nozzle-plate distances and pressure ratios were experimentally investigated. The results suggested that all the flow fields could be classified into three types of flow structure (See Fig. 4) [8]. However, flow structure details were not clarified because analysis was based on the schlieren pictures and the pressure contours over the flat plate obtained from PSP images.

Simulations with conventional method (RANS simulations with a second-order TVD upwind scheme together with a well-known turbulence model) were consequently carried out [9]. Almost fifty cases with different geometrical and physical parameters were simulated and the results were discussed. In Fig. 5, Pressure distributions at the pressure ratio PR=7.4 for four different plate angles and four different plate distances are shown as an example. There observed are several types of pressure peaks. Some cases have single peak and some cases have a few different type of peaks. Figures 6 shows an example of the flow structure details found through the analysis of the simulation results. Flow structure and the associated pressure peaks are much better understood when the flow field is analyzed using whole the three-dimensional data for many cases.

Second factor: LES and LES/RANS hybrid methods

There is an obvious shift in the CFD research from RANS simulations to Large Eddy Simulations (LES). The shift is supported by the rapid progress of computer performance, but more importantly, we start to recognize that the nature of flow physics, even from the engineering viewpoint, require unsteady flow simulations. Separated flows are inherently unsteady and steady recirculating region may be the result of a time average of strongly unsteady flows. Capturing such unsteady flow behavior is inevitable for the analysis and eventual control of the flows.

Requirement of the grid numbers for LES becomes enormous. In addition, fine mesh resolution near the wall boundaries limits the time step size for the computation. Therefore, it still remains difficult to apply LES to complex flows at high Reynolds numbers. To

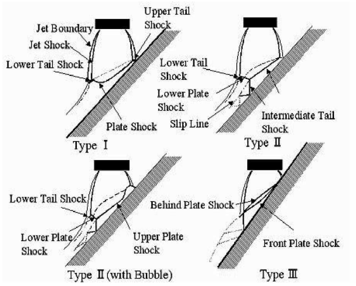

Fig. 4. Schematic pictures of typical flow fields for various plate-angle(PR>4)

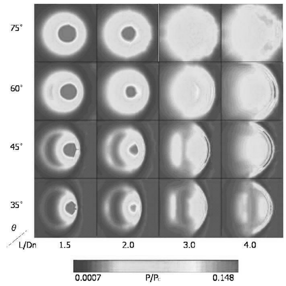

Fig. 5. Pressure contours on the plate surface: PR 7.4

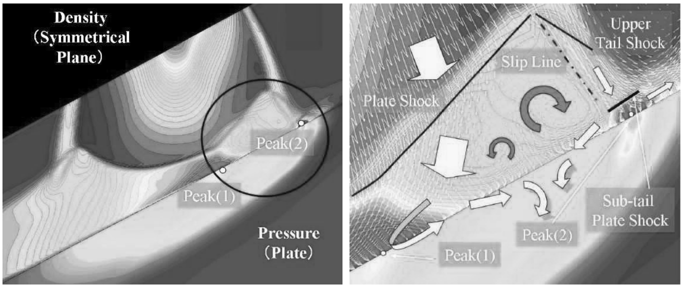

Fig. 6. Mechanism of the pressure peak (2) ($PR = 7.4$, $L/Dn = 3.0$, $\theta = 60°$)

overcome these difficulties, LES/RANS hybrid methodology was proposed, where 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 require much less computational cost than LES as it alleviates the required mesh resolution near (and along) the walls and the resultant time-step limitation.

As an example of this approach, simulation of supersonic base flow is presented [10]. Accurate simulation of base flows is important as the base drag influences aerodynamic characteristics of the vehicle at certain speed range. As shown in Fig. 7 schematically, supersonic base flow includes a large recirculation region and interaction of shear layer with expansion and compression waves appears. Even with such practical importance, pressure distributions over the base were not well estimated in any simulation using RANS models.

Figures 8(a) and 8(b) show instantaneous views of the computed vorticity magnitude contours computed by the LES/RANS hybrid method and RANS method. There exists strong flow unsteadiness in Fig. 8(a), whereas no unsteadiness is observed in Fig. 8(b). When time-dependent results computed by the LES/RANS hybrid method is averaged in time, vorticity magnitude contours become similar to Fig. 8(b) except the size of recirculating region. This indicates that steady flow field typically known is a time-average of the flow field with strong unsteadiness. The time-averaged base pressure distributions along the base surface are compared with the experiment in Fig. 9. The RANS result shows a strong variation of the pressure distributions due to a strong reverse flow, whereas LES/RANS hybrid method shows a flat distribution which agrees well with the experiment.

The results above show that capturing the unsteady nature of flow field leads to accurate flow simulations and the LES/RANS hybrid method including DES method is appropriate for accurate simulation of complex flows under reasonable computer resources. The LES/RANS hybrid method is a powerful method but at the same time has limitations, which we should keep in mind. One obvious deficiency is flow transition. Developing a method to capture "Scale Effects" is an important but still unresolved issue in CFD [4]. A series of flow simulations at subsonic, transonic to supersonic conditions were later carried out, resulting in better understanding of physics of cylindrical base flows [11].

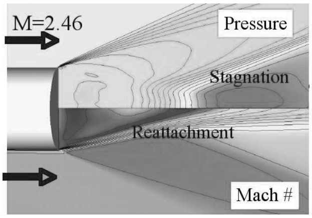

Fig. 7. Feature of supersonic base flows

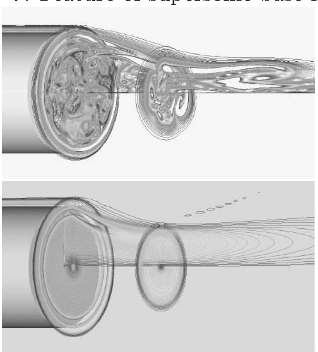

Fig. 8. Instantaneous view of the vorticity magnitude contours: (a) LES/RANS hybrid (top), (b) RANS (bottom)

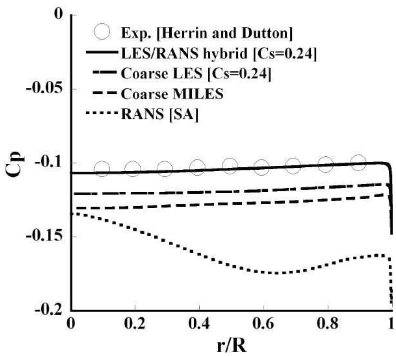

Fig. 9. Pressure distributions in the radial direction at the base

Third factor: Spectral-like schemes

Recently, compact difference scheme receives people's attention. The scheme uses Pade approximation, the idea of which is not new, but with the high-order filtering technique, Lele [12] showed compact difference scheme has a potential for practical simulations. Implementation in the generalized coordinate systems and real applications shown by Gaitonde and Visbal [13] expanded the use. Other compact schemes also appeared as their derivatives. The spectral-like schemes reduce the number of grid points, for instance, necessary for capturing vortex structure. CFD researchers usually consider that 20 to 25 points are necessary inside one vortex but the number is reduced a few times for each direction when the compact scheme is used. In three dimensions, reduction of required computer memory and computer time becomes enormous.

One important shortcoming of the compact scheme is that it cannot be applied to simulations of flows with shock wave. Nonlinear formulation is required to capture discontinuities and various non-linear schemes, such as Weighted Essentially Non Oscillatory (WENO) scheme [14] and Weighted Compact Non-linear Scheme(WCNS) [15] were developed. Also effort to improve filtering techniques in the original compact scheme is pursued [16]. Among them, we consider that an explicit version of WCNS is efficient and still keeping certain level of high resolution of the flow features applicable to practical simulations for shock-related problems [17].

Acoustic noise generated by the impingement of an over-expanded supersonic jet on a flat plate is of practical importance in many engineering applications. Rocket launching is an example. Both satellite and rocket vehicle are exposed to severe structural environment due to acoustics from rocket plumes. Such a plume structure including shock cell and stand off shock interaction with jet shear layers is physically complicated, and such flow structures determine characteristics of emanated acoustic waves. There also exists interaction of plume flows with the complex launcher structures. There is a well-known model of rocket plume acoustics proposed by NASA SP-8072 [18], in which acoustic load over the vehicle is estimated using a semi-empirical method based on the existing numerous experimental data. The model gives us easy and fair estimation method but they usually require empirical factors adjusted to particular launchers, and extension to new launching sites is not easy. Better estimation method would reduce the acoustic and vibration tests of satellites, which may increase the weights of satellites and shorten the period of satellite development. There are many experimental and numerical reports available for plume acoustics but particular attentions were paid to discrete and intensive tones rather than broadband tones. Rocket plume acoustics are not well understood.

As a practical problem of helping the design of JAXA's launching sites, we conducted simulations of plume flows for real rocket and launching site configurations. So far, we only used conventional second-order upwind shock capturing method. As shown in Fig. 10, compression waves reflect over the launcher surface and the tower, and go up toward the fairing of the rocket. Acoustics emit mainly from the plume mainly after the potential core (supersonic region inside the plume).

Simulation using 7th-order WCNS scheme for a model problem is tried in parallel [19]. Primary purpose is to understand the aero-acoustic mechanisms of over-expanded supersonic jet impinging on a flat plate with and without holes. For finding the required grid resolution for capturing sound radiations from the jet and finding important phenomenon of plume acoustics, two-dimensional axisymmetric flow simulations were carried out. Obviously, axisymmetric flow assumption is not valid, but effectiveness of WCNS scheme is discussed as our first step. Figure 11 shows the pressure contour plots of the model problem of rocket plume from a single solid motor impinging on the launcher with a hole. Strong compression waves emanate from certain area downstream of the plume with a strong directivity. Phenomenon is same as in Fig. 10, but reflected compression waves are better captured in this computation by the WCNS scheme.

4 Requirements for New Type of Simulations

Progress of computer hardware has been a major factor that accelerated CFD research since supercomputers appeared in late 1970's. Speed of leading-edge supercomputers

Fig. 10. Pressure contours of rocket plume simulations (RANS simulation)

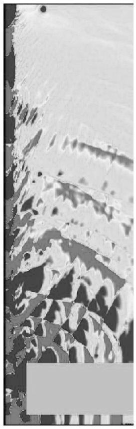

Fig. 11. Pressure contours: model probem

increases almost twice every year[4]. Now, Japan plans to develop a class of 10 PFLOPS supercomputer in 2011 and U. S. may have a similar plan. Performance of such leading-edge computers would be roughly ten to hundred million times faster than the first commercial supercomputer that first appeared late 1970's. In Table 1, author's personal estimation of required computer time and memory is presented. The estimation is based on the assumption that simulation requires LES with a spectral-like scheme. The number is based on a rough estimation that such a scheme would reduce the required memory and cpu time about 100 times smaller than conventional second-order schemes. The numbers obviously depend on many factors such as computer architecture and others. From the estimation, simulations of flows at 10^5 order of Reynolds number will become feasible soon and even LES simulations at higher Reynolds numbers may become feasible when Petaflops computers become available.

Table 1. Number of grid points and computer time required for spectral-like methods

	Lower than $Re = 10^5$ MAV, UAV, Mars Aircraft	$Re = 10^6$ Wind tunnel level	Higher than $Re = 10^7$ Civil transports
No. Grid points (required memory size)	10^7 (10 GB)	10^9 (1 TB)	10^{11} (100 TB)
Computational time on 1 TFLOPS computer	10-50 hours	1,000-5,000 hours	100,000-500,000 hours

5 Final Remarks

Computational Fluid Dynamics (CFD) contributed to analysis of high-speed shock-related problems extensively. Some of the study examples by the author's group in the past were presented, which showed effectiveness of CFD both for design of transportation vehicles and for understanding of fluid flows. Trend of CFD for further use was discussed based on the recent effort in the CFD community and its applications. There are three key features: computer progress, spectral-like high-resolution scheme, and LES and LES/RANS hybrid method. Recent studies revealed that high-speed flows showing steady state features in the experiment are inherently unsteady and that requires LES-like computations for successful simulations. Such simulations require very high grid resolution, and spectral-like high-resolution scheme would help reducing the number of grids and computer time necessary. With these emerging techniques, CFD will have another prospect and hopefully will contribute much more to high-speed shock-related problems.

References

1. Murman, E. M. and Cole, J. D., Calculation of Plane Steady Transonic Flows, AIAA J., Vol. 9, pp. 114-121, 1971
2. Lax, P. D., Weak Solutions of Nonliner Hyperbolic Equations and their Numerical Computation, Comm. Pure Appl. Math., Vol. 7, pp. 159-93, 1954
3. Shang, J. S., A Glance Back and Outlook of Computational Fluid Dynamics, ASME FEDSM2003-45420, 2003
4. Fujii, K., Progress and Future Prospects of CFD in Aerospace-Wind Tunnel and Beyond, Progress in Aerospace Sciences, on International Review Journal, Vol. 41, pp. 455-470, 2005
5. Ogawa, T. and Fujii, K., Prediction and Alleviation of a Booming Noise Created by a High-speed Train Moving into a Tunnel, The ECCOMAS Computational Fluid Dynamics Conference, 1996
6. Fujii, K. and Ogawa, T., Aerodynamics of High Speed Trains Passing by Each other, Computers & Fluids, Vol. 24, pp. 897-908, 1995
7. Ogawa, T. and Fujii, K., What Have We Learned from CFD Research on Train Aerodynamics, Frontiers of Computational Fluid Dynamics 1998, Ed. by D. A. Caughey and M. M. Hafez, World Scientific, November 1998
8. Nakai, Y., Fujimatsu, N. and Fujii, K., Experimental Study of Underexpanded Supersonic Jet Impingement on an Inclined Flat Plate, AIAA J. Vol. 44, pp. 0001-1452, 2006
9. McIlroy, K. and Fujii, K., Computational Analysis of Supersonic Uinderexpanded Jets Impinging on an Inclined Flat Plate, AIAA Paper 2007-3859, June 2007
10. Kawai S. and Fujii K., Computational Study of a Supersonic Base Flow Using Hybrid Turbulence Methodology, AIAA Journal, Vol. 43, 2005, pp. 1265-1275, 2005
11. Kawai S. and Fujii K., Time-series and Time-Averaged Characteristics of Subsonic and Supersonic Base Flows, , AIAA J., Vol. 45, pp. 289-301, 2007
12. Lele, S. K., Compact Finite Difference Schemes with Spectral-Like Resolution, J. Comp. Phys., Vol. 103, pp. 16-42, 1992
13. Gaitonde, D. V. and Visbal, M. R., Further development of a Navier-Stokes solution procedure based on higher-order formulas, AIAA Paper 99-0557, 1999
14. Liu, X.D., Osher, S. and Chan, T., Weighted essentially non-oscillatory schemes, J. Comp. Phys., Vol. 126, pp. 200-212, 1996
15. Deng, X. G. and Zhang, H., Developing high-order weighted compact nonlinear schemes, J. Comp. Phys., Vol. 165, pp. 22-44, 2000
16. Fiorina, B. and Lele, S. K., Artificial nonlinear diffusivity method for supersonic reacting flows with shocks, J. Comp. Phys., Vol. 222, pp. 246-264, 2007
17. Nonomura, T. and Fujii, K., Increasing order of Accuracy of Weighted Compact Non-Linear Scheme, AIAA Paper 2007-893, Jan. 2007
18. Eldred, K. M., Acoustic Loads Generated by the Propulsion System, NASA SP-8072, June 1971
19. Kawai, S., Tsutsumi, S., Takaki, R. and Fujii, K., Computational Aeroacoustic Analysis of Overexpanded Supersonic Jet Impingement on a Flat Plate with/without Hole, FEDSM2007-3756, 5th Joint ASME/JSME Fluid Engineering Conf., July 2007

http://www.springer.com/978-3-540-85168-4

Shock Waves
26th International Symposium on Shock Waves, Volume
1
Hannemann, K.; Seiler, F. (Eds.)
2009, XXX, 799 p.,
ISBN: 978-3-540-85168-4