THE APPLICATION OF THE NUMERICAL SIMULATION IN CAE

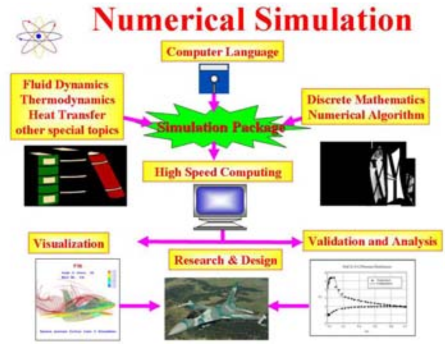
Chang-Hsien Tai, Ph.D.
Vice President for Academic Affairs
Professor of Department of Vehicle Engineering
National Pingtung University of Science and Technology

ABSTRACT

CFD is the simulation of fluid engineering systems using modeling (mathematical physics problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.) In recent years, the progress in computer software, hardware and numerical methods have greatly improved. Numerical simulation models have been applied extensively to CAE (Computer-Aided Engineering). In numerous research and development areas contributions have been made in vehicle design, plane design, weather forecasting, weapon systems, electronic products and building structures. Numerical simulations can reduce the risk and cost in experimental testing, reduce the developmental time period and permit the visualization of physical fluid phenomena that are difficult in experimentation. NPUST NSLab, Numerical Simulation Lab. originated in national defense research. We have applied these studies to the civil industry in recent decades The achievement results have been substantial.

I. What is CFD?

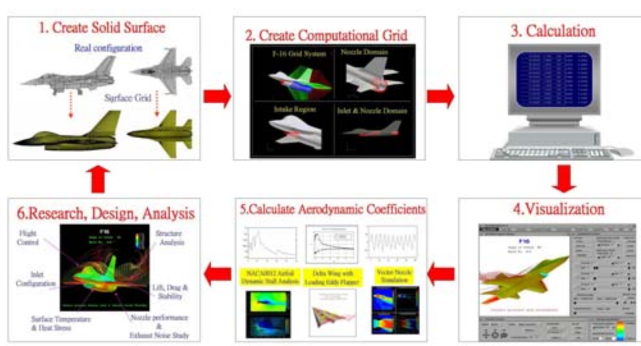
Computational Fluid Dynamics or CFD is used to generate computer flow simulations. CFD involves numerical solutions for the laws governing fluid dynamics. The complex sets of partial differential equations are solved in the geometrical domain divided into small volumes, commonly known as a mesh (or grid). The basic CFD sciences consist of mathematical rules, computer science and other relevant professional technology, as shown in fig. 1. CFD has enabled us to understand the world in new ways. We can now see what it is like to be in a furnace, model how blood flows through our arteries and veins and even create virtual worlds. CFD enables analysts to simulate and understand fluid flows without the help of instruments for measuring various flow variables at desired locations.



II. CFD Procedure

CFD codes have different purposes for different applications: investigation of bubble-fluid interactions for bubbly flows, study of wave induced massively separated flows for free-surface, etc. Depending on the specific problem and flow conditions, different CFD codes can be chosen for different applications (aerospace, marines, combustion, multi-phase flows, etc.) Once the CFD purpose is chosen, a "CFD process" is determined for the steps necessary to set up the IBVP problem (Initial Boundary Value Problems) and run the code, as shown in fig.2:

1. Geometry: Geometry is usually created using commercial software. Determine the domain size, shape and simplifications (smooth over the small steps).
2. Mesh: Flow conditions and fluid properties. Meshes should be well designed to resolve the important flow features that are dependent upon flow condition parameters. A mesh can be generated using either commercial codes or research code.
3. Physics and Solve: Select the models and the initial boundary conditions. Solve the momentum, pressure Poisson equations and obtain the flow field quantities, such as velocity, turbulence intensity, pressure and integral quantities
4. Post-processing: Post-processing visualizes the CFD results (usually with commercial software). The simulation data can be imported for validation.
5. Report: Reports save the time history of the velocity, pressure and temperature, etc. Report the integral quantities, such as total pressure drop, total friction force, friction factor, lift and drag coefficients.
6. Analysis: Using the report to design or improve the product.

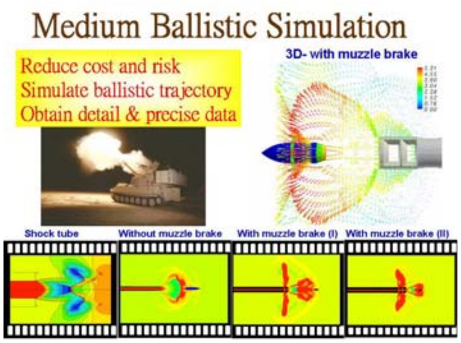


III. The application and benefit of the numerical simulation technology

The national defense industry was the earliest social organization to use numerical simulation technology. Defense industry use included defense-vehicle design, weapon development, jet engines, war games, weather forecasting, education, manufacturing, C4ISR, etc. There are many successful examples of numerical simulation design use to reduce cost in the world. For example, the Tu-22 bomber in Russia used simulation to lower cost and obtain the maximal benefit. The TU-22 bomber was in service at that time. The wing aerodynamics were modified using numerical simulations and experiments. The results promoted Russian air force capacity.

The early experimental designs used numerical simulation technology in research, development, design and processing of advanced USAF fighter jets. Numerical simulation has recently brought another kind of revolutionary development to industry as follows,

1. Low cost, efficiency, and safety in experiments with dangerous military systems Such as the calculating a bullet's trajectory as shown in fig.3. The traditional way must launch as many as a million bullets,



accumulate and calculate through experiments to develop a useful firing table.

2. Validate the experimental result using real phenomenon: an error exits between the numerical simulation and real physical phenomenon. However, error within a certain limit is allowed in industry depending on the target application.

3. Not influenced by the size of the experiment: capable of modeling minute systems such as individual electrons or massive systems such as cargo airplanes. The numerical simulation is a good tool with low cost and high efficiency.

4. Detailed data information without experiment technology limit: Such as entering plane experiment or test stage, there are numerous aerodynamic coefficient correlations (ex: burning in combustor, high temperature heat transfer and lift and drag force at high AOA, etc.) It is difficult to get detailed information using experimental methods or actual tests. For this reason, numerical simulations provide detailed test data with good criteria for validation or improvement.

5. Shorten the development period in engineering: in the Boeing Company from 1990, a new airplane design applied numerical simulations for the airframe design, power plants, landing gear system, etc...

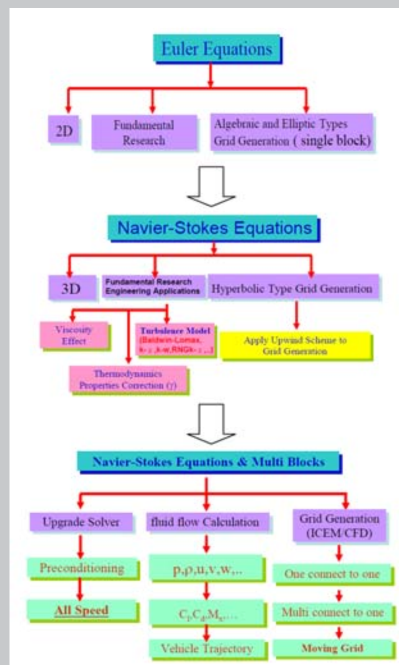
IV. Introduction of NPUST NSLab research results in recent years

1. All Speed and High-Resolution Scheme Development

Numerical flow solver use has expanded throughout industry, becoming necessary in computational fluid dynamics (CFD) methodologies that must operate robustly and efficiently over a wide range of Reynolds numbers and Mach numbers. To cope with this demand, modern computational methods tend to be general-purpose methods covering flow regimes from low speed or incompressible to transonic and supersonic flow in a variety of complex 3-D geometries. Most numerical methods have the aim of solving incompressible flow problems with numerical algorithms that were designed for compressible flows. Preconditioning is essential when time-marching methods are employed for low Mach number compressible or incompressible problems because of the eigenvalue stiffness inherent under these conditions. In addition to convergence and round-off error difficulties, the solutions for compressible fluid flow equations suffer from accuracy problems in the low Mach number limit. Over the past years, several authors have investigated and shown that the time-derivative preconditioning accelerates the convergence to a steady state for solving the Euler and Navier-Stokes equations.

The application of preconditioning changes the eigenvalues in the system of compressible flow equations and reduces this disparity in the wave speeds. The time derivatives are pre-multiplied by a matrix that slows the speed of the acoustic waves down towards the fluid speed. Hence, at low Mach numbers all of the waves will have the same order and the system will be well-conditioned. Using proper preconditioning leads to a numerical scheme that behaves appropriately for low Mach numbers. As the preconditioning provides a very promising tool to solve gas dynamic problems over a wide range of flow speeds, grid generation is a very crucial substep in the global numerical simulation process. Because the accuracy of computed results depends directly on domain discretization, single structured grids are not sufficient in many realistic applications while multi-block grids provide great flexibility in discretizing geometrically complex domains.

In this study, we describe the AUSM family of low-diffusion flux-splitting methods developed by Liou to operate effectively in conjunction with time-derivative preconditioning. The eigenvalues of the preconditioned system are used directly to affect the transition from a formulation suitable for high-speed flows to one suitable for low-speed flows and modify the time steps to ensure a stable scheme.

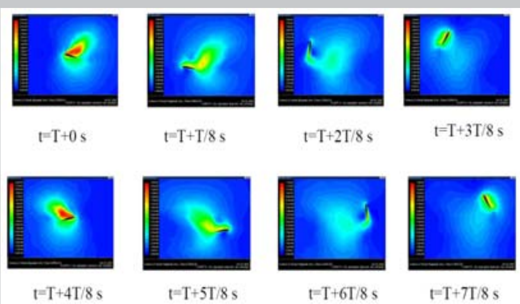


Developing process of the self–developed code, HASP

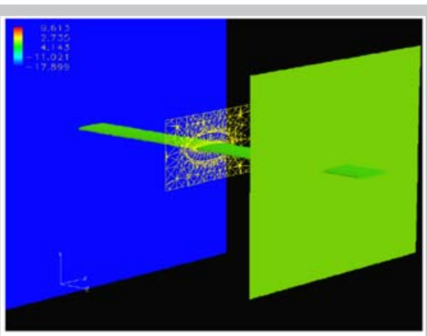
2. Ornithopter

Future aircraft, however, may fly more like birds, by properly adapting the geometries of their wings to suit changing flight conditions. The Ornithopter is a flapping aircraft like nature's flyer. Since a flapping wing combines lift and thrust, its' maneuverability and control performance are superior to traditional man made aircraft. However, the aerodynamic features of an Ornithopter are quite different from the traditional plane, such as lower aerodynamic efficiency; high sensitivity to turbulence and unsteady phenomena (ex: delayed stall, dynamic stall).

In this study, we will use the Ornithopter wing to perform fundamental research for morphing wings. We use the CFD method to solve the flapping and hovering problems. The governing equation is the fully elliptical, three-dimensional Reynolds-averaged Navier-Stoke's equations. In order to probe the detail vortex dynamics and predict the force for hovering and flapping, a sufficient algorithm and dynamic grid system is required. Therefore, the mesh used in the finite-volume numerical computation is the multi-block, hybrid, dynamic and moving grid system. The vortices phenomena of the delayed and dynamic stall for a hummingbird during the figure-eight motion is investigated. There are two validation cases simulated, as shown in this study. The first uses the 2D elliptical wing to simulate the flight of dragonflies. The second is the flapping wing. Comparing the result with literature data, the selected numerical method is applied to the figure-eight motion of a hummingbird. The numerical results show that the Ornithopters depend on vortices to keep them aloft, especially when they are hovering. The vortex shedding is due to a rapidly oscillating wing at a low Reynolds number. The translation delayed stall occurs when a wing at an angle of attack well above the stall angle starts moving from rest.

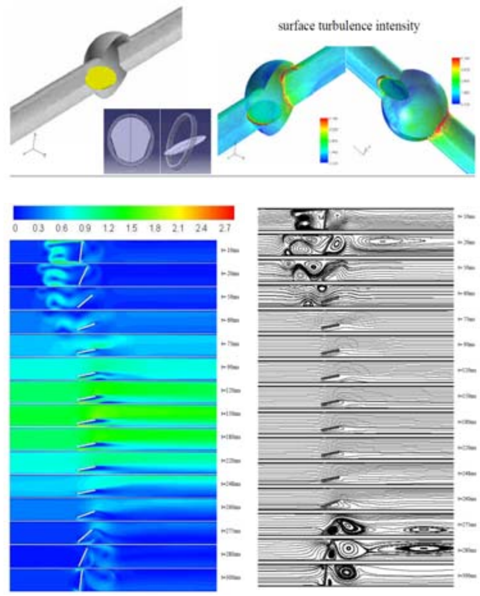


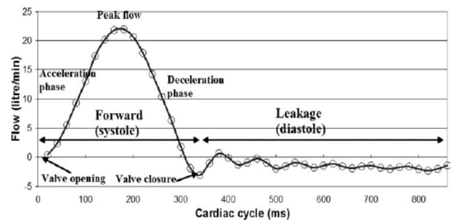
Vortices field illustrate the formation and dynamics during hovering


3D flexible flapping wing

3. Tilting-Disc Heart Valve in Pulsatile Channel Flow

A fully-coupled fluid-structure interaction software system for a pulsatile flow across a moving tilting-disc valve with clearance is developed. Unsteady pulsatile flow coupled with induced valve motion has been examined in detail. State-of-the-art computational fluid dynamic (CFD) methods are adopted in the present flow solver development. The incompressible finite-volume method is employed to solve the Reynolds-averaged Navier-Stokes equations in conservation form. To account for nonstationary valve motion, the meshes surrounding the valve are generated and updated in each time-marching step using the hybrid grid method. A single-degree-of-freedom rotational valve model is integrated simultaneously with the CFD adaptive time-stepping. It is found in the present study that, on both side of the occluder, strong shedding vortices occur and persist in the valve closing phase. These closure vortices show great influence on the prediction of the regurgitate flow characteristics and the subsequent valve opening dynamics as well. Based on the present software system, the results obtained from quasi-steady simulations performed at various instants of interest with prescribed valve motion are critically evaluated to assess whether simplified flow and valve conditions may lead to erroneous conclusions.





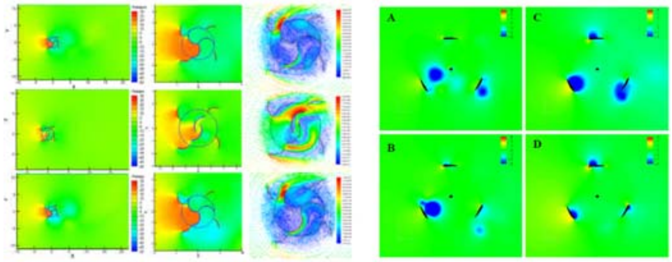
4. Dynamic Simulation of a Vertical Axis Wind Turbine

There are two main categories of wind turbines: Horizontal Axis Wind Turbines (HAWT) and Vertical Axis Wind Turbines (VAWT). HAWT designs are currently widely used, but have the disadvantage that they have to be positioned perpendicular to the wind direction. VAWT have the advantage that they can be positioned independent of the wind direction.

Efficiency remains the main problem with VAWT. Its function mainly produces the inertial force of the pulsatile moment reduced by the lift to drive the wind turbine to finish periodic motion, but the negative moment reduces efficiency. Consequently, one method to improve wind turbine efficiency is to reduce the negative moment or increase the positive moment. This is the important reference via torsion analysis in the wind turbine. Because of no optimal wind blade design, the output efficiency is poor. This study uses the unsteady 2nd-order upwind implicit SIMPLE algorithm, sliding grid, and coupling a user defined subroutine of the degree freedom dynamic method as the solver to simulate the on wind load fluid-structure interaction (FSI) flowfield of a vertical axis wind turbine (VAWT), and analyze the aerodynamic characteristics.

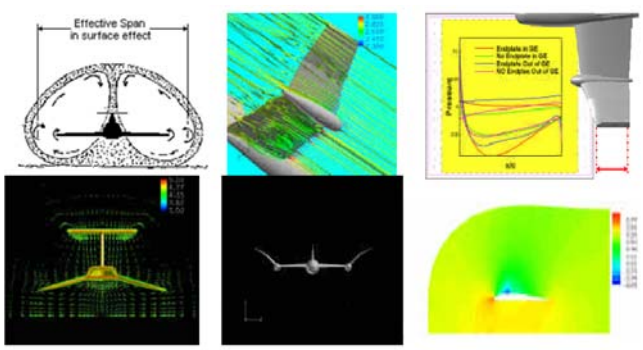
5.The Research on WIG

WIG is unique to the class of vehicle de



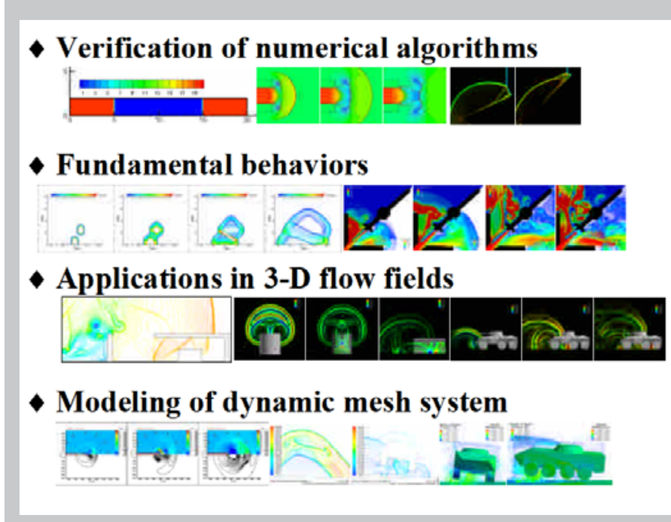
signed to fly very close to a surface (water or ground), and that in so doing these vehicles benefit from the peculiar aerodynamics in that region. The proximity of the surface both "compresses" the air under the wing; giving it more lift and reduces the drag from the wig, making it more efficient. This effect increases as the surface gets closer and is the basis for self-stabilization in the "ground effect". In concept, such craft offer a potential form of transportation that is faster than boats and slower than aircraft. They also offer greater fuel economy and the potential to carry heavier loads than aircraft. The main issues to be resolved are seaworthiness and safe operation.

The flow structure within the passage, between the wing and boundary, is very complex and difficult to predict. Laminar flow, transitional flow, turbulent flow and the secondary flow caused by the span wise pressure gradient take place at the same time. In some cases, there will be a separation bubble on the airfoil's suction side or pressure side. According to the results of this research, this analyzer can indeed effectively modulate and simulate the aerodynamic characteristic of the WIG.



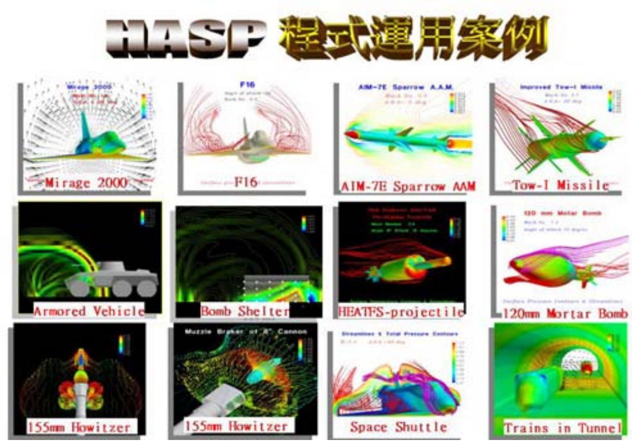
6. Investigation of a Blast Wave impacting on Buildings and Vehicles

The physical phenomena associated with a blast wave has been studied for more than half a century. The complex shock-shock and shock-vortex interaction phenomenon that takes place following the detonation of an explosive device includes reflection, diffusion and consolidation events. Researchers have utilized both experimental and numerical approaches in an attempt to clarify the physics of the blast phenomena and for developing techniques to predict the likely effects of blast waves. This study investigates the behavior of blast waves by employing the finite volume method to solve the associated three-dimensional, time-dependent, inviscous flow Euler equations. The complex transitional shock phenomena are investigated using a multi-block mesh system and a flux computational model. The Total Variation Diminishing (TVD) upwind method is applied. Spatial discretization is performed using Roe's solver with high-order Kappa Monotone Upwind-centered Schemes for Conservation Laws (MUSCL) interpolation. Time integration is achieved via the second-order explicit Hancock method. This study has also developed a code that combines fluid and acoustic simulations.



7. Others

Numerous EI/SCI journal papers have been published on MHD, air-conditioning, vehicle, missile, submarine, computer cooler, MAV, UAV, two phase flow, MPI, etc.



V. Conclusion

The domestic industry is in a transformation period. The government will lead Taiwan towards a Regional Science and Manufacturing Center in the Asia-Pacific region. Every domestic industrial producer should possess individual design capability. However, numerical simulation application still not as good as that of American or European countries in project design and application. If we improve the production efficiency in product research and development, we will certainly combine the talents and resources of domestic universities, industry technology. CAE/CAD/CAM and CFD must expand the range of research and development. Numerical simulations must be used before the preliminary design. Successful foreign experiences must be studied. Our industrial engineering level can then attain the world-class realm.