Close out and Final report for NASA Glenn Cooperative Agreement NCC3-748

Advanced Turbulence Modeling Concepts

Tsan-Hsing Shih

January 2005

Summary of Research During Tenure

1. CFD STUDIES FOR ZERO CO2 EMISSION TECHNOLOGY (ZCET) PROGRAM

The ZCET program developed at NASA Glenn Research Center is to study hydrogen/air injection concepts for aircraft gas turbine engines that meet conventional gas turbine performance levels and provide low levels of harmful NOx emissions.

A CFD study for ZCET program has been successfully carried out. It uses the most recently enhanced National combustion code (NCC) to perform CFD simulations for two configurations of hydrogen fuel injectors (GRC- and Sandia-injector). The results can be used to assist experimental studies to provide quick mixing, low emission and high performance fuel injector designs.

The work started with the configuration of the single-hole injector. The computational models were taken from the experimental designs. For example, the GRC single-hole injector consists of one air tube (0.78 inches long and 0.265 inches in diameter) and two hydrogen tubes (0.3 inches long and 0.0226 inches in diameter opposed at 180 degree). The hydrogen tubes are located 0.3 inches upstream from the exit of the air element (the inlet location for the combustor). To do the simulation, the single-hole injector is connected to a combustor model (8.16 inches long and 0.5 inches in diameter). The inlet conditions for air and hydrogen elements are defined according to actual experimental designs. Two crossing jets of hydrogen/air are simulated in detail in the injector. The cold flow, reacting flow, flame temperature, combustor pressure and possible flashback phenomena are studied. Two grid resolutions of the numerical model have been adopted. The first computational grid contains 0.52 million elements, the second one contains over 1.3 million elements. The CFD results have shown only about 5% difference between the two grid resolutions. Therefore, the CFD result obtained from the model of 1.3-million grid resolution can be considered as a grid independent numerical solution.

Turbulence models built in NCC are consolidated and well tested. They can handle both coarse and fine grids near the wall. They can model the effect of anisotropy of turbulent stresses and the effect of swirling. The chemical reactions of Magnusson model and ILDM method were both used in this study.

These CFD studies of single-hole injectors provide useful information about hydrogen/air mixing, its effect on reacting flow structures and NOx emission, etc. The performance of the single-hole injector affects the performance of the whole combustor system. Following the CFD simulations of the single-hole injector, the CFD simulation of the whole combustor system will start by using the CFD results of the single-hole injector to extrapolate the inlet conditions for CFD simulations of the whole combustor system. This will be a continued research project.

2. ADVANCED TURBULENCE SIMULATION FOR PROPULSION SYSTEMS

- Three-dimensional unsteady simulation of complex flows in propulsion systems using NCC.

Flows in propulsion systems are complex, often unsteady. Flow separation, swirling, high rate of shearing, etc. are likely unsteady. This unsteadiness may have great importance to the performance of the propulsion systems. Therefore, an accurate 3-D unsteady simulation methodology of NCC for complex reacting flows must be established. This will involve turbulence model, chemistry model, hybrid grid generation of hexahedral, pyramid and tetrahedral. Several benchmark flows have been used to serve this purpose.

- Gas turbine combustor (GE lean premixed dry low-NOx LM6000). This combustor comprises of a rectangular box with two blocks located at top and bottom surfaces from which cooling air is blown downstream. A highly swirling jet is injected from a circular inlet under high pressure and temperature conditions. The experimental data and LES data are available for comparison.

- Backward facing-step flow, which is a typical flow separation benchmark flow. DNS and experimental data are available.

- 3-D cylinder vortex shedding flow, which is a traditional unsteady benchmark flow. Experimental data are available.

Both steady and unsteady calculations using Reynolds Averaged Navier-Stokes (RANS) model have been performed and compared with existing experimental and DNS data. The results showed partial success of RANS model for relative weak shear and relative weak swirling flows.

- NCC Large Eddy Simulation of complex propulsion flows.

Many researchers have found that the highly unsteady turbulent reacting flows with large scale structures, such as separation, recirculation, swirling and strong shear, etc. are very difficult to be accurately modeled by Reynolds Averaged Navier-Stokes equations (RANS). We have had similar experience in our current studies described in Section A. It is suggested by many researchers that the method of large eddy simulation (LES) may provide a realistic solution to these flows. NASA Glenn Research Center and Georgia Institute of Technology are collaborating to develop this technology in NCC. This will involve all levels of sub-grid scale (SGS) stress models, accuracy of numerical schemes and proper 3-D hybrid grid generation. The followings have been conducted for development of NCC-LES:

- Study the effect of 2nd and 4th order numerical dissipations (built in NCC) on the numerical solution using the flows of (a) cylinder vortex shedding, (b) GE LM6000 dump combustor. We have successfully managed to set the 2nd order dissipation to zero and a minimum value of 4th order dissipation for unsteady calculations. This provides a meaningful numerical base for LES.

- Establish a procedure of generating an initial flow field for LES of LM6000 combustor.

- Study of inlet and outlet boundary conditions for LES of internal flows.

  1) Carry out the LES of GE-LM6000 gas turbine combustor using various LES sub-grid models, including zero-equation Smagorinsky model, one-equation Menon's model, two-equation k-eps model. This will demonstrate the capability of NCC-LES for complex turbulent flows.
  2) Develop a module of NCC-LES, which contains Smagorinsky zero-equation model, one-equation k model and two-equation k-eps model. This will enable NCC for a broad range of applications in propulsion systems. Users can chose a model scheme for their particular problem from various turbulence model schemes, including steady/unsteady RANS models, LES models.

- Flow physics, modeling and NCC enhancement

  To support NCC-LES of complex reacting flows for propulsion systems, a suitable method of large eddy simulation must be developed. Since NCC code is a second order accuracy scheme we must ensure that the sub grid scale (SGS) stresses dominate over the numerical dissipation. This will require a larger value of SGS stresses (corresponding to a larger resolved length scales of turbulence) in order to reduce the contamination of the numerical dissipation. For this reason, a two-equation (turbulent kinetic energy and dissipation) SGS model has been adopted. This is particularly important for high Reynolds number turbulent flows, because for a given resolved (larger) scale, the extent of turbulence modeling increases with the Reynolds number and that will require more accurate SGS models. The calculations using a two-equation SGS model have been successfully carried out for LM6000 dump combustor.

3. ADVANCED TURBULENT COMBUSTION MODELING CONCEPTS

- Development of a new strategy "Partially Resolved Numerical Simulation" for high Reynolds number complex combustor flows using NCC.

  - Reviewed existing traditional LES methodology: subgrid models, dynamic procedures, requirement of numerical platform, possible engineering applications for practical gas turbine engines. The conclusion is that the traditional LES is very limited for practical engineering applications due to the following reasons: it requires the fine mesh spacing, it is limited to the moderate Reynolds number flows, it requires high order accuracy or "zero dissipation" CFD codes, the solution is in general grid dependent.

- To overcome the shortcomings of the pure LES, we proposed a new strategy called Partially Resolved Numerical Simulation (PRNS). The resolved large-scale turbulence is defined using the temporal filtering instead of the spatial filtering. The unresolved turbulence is grid independent and is modeled grid-independently. Mesh spacing could be quite coarse as long as it can resolve the resolution of large-scale structures. It can handle very large Reynolds number flows with more or less the same mesh spacing since the resolution of turbulent large-scale structures is more or less independent of the Reynolds number. We are aiming to handle a real size engine analysis with about 5 million grid points for high Reynolds number combustor flows. In our validation case of LM6000 dump combustor, only 0.5 million grid points were use in a successful PRNS simulation.

- Programs for data analysis: one- and two-point turbulent correlations, power spectrum density (PSD) for turbulent quantities, etc. are developed for various data analysis.

- Validation of PRNS:

  Validations and applications of PRNS were successfully carried with the following two types of turbulent flows:

  - General Electric LM6000 gas turbine combustor. The flow is complex and represents the typical combustor flow structures: massive separation, swirling and recirculation. We have demonstrated that 1) the PRNS strategy only needs less than 0.5 million grid points to obtain LES type of flow structures. 2) the implementation of PRNS is very simple for any turbulent CFD code that can run time dependent calculations.

  - Pipe flows at various Reynolds numbers. The flow geometry is very simple but is of rich turbulent characteristics. Many experimental data for a wide range of Reynolds numbers are available; a few DNS data are also available in the literature. These data can be used to validate the performance of PRNS strategy.

4 PUBLICATIONS:

1) Shih, T.-H., Norris, A., Iannetti, A., Marek, J., Liu, N.-S., Smith, T., and Povinelli, L.A., "A Study of Hydrogen/Air combustor Using NCC," AIAA 2001-0808, 2001.
2) Shih, T.-H., Povinelli, L.A. and Liu, N.-S., "Application of generalized wall function for complex turbulent flows," ETME 5, pp177-187, Mallorca, Spain (2002).
3) Shih, T.-H., Smith, T., Marek, C., Iannetti, A., Norris, A. and Liu, N.-S., "Numerical Study of a Single Hydrogen/Air Cas Turbine Fuel Nozzle," AIAA-2003-4249, 33rd AIAA Fluid Dynamics Conference and Exhibit Orlando, Florida 23-26 June 2003.
4) Shih, T.-H. and Liu, Nan-Suey, "Partially Resolved Numerical Simulation: from RANS towards LES for Engine Turbulent Flows," AIAA -2004-0160.

5) Shih, T.-H. and Liu, Nan-Suey, "A Unified Strategy for Numerical simulation of Turbulent Flows," Internal Report, Oct. 2004.
6) Liu, Nan-Suey and Shih, T.-H., "From Reynolds Averaged Navier-Stokes Simulation towards Large Eddy Simulation" submitted to J. of AIAA, November, 2004.