**LES SOFTWARE FOR THE DESIGN OF LOW EMISSION COMBUSTION SYSTEMS
FOR VISION 21 PLANTS**

Quarterly Technical Progress Report for

July - September 2003

by

Clifford Smith

September 2003

CFDRC Report No. 8321/12

Contract No.: DE-FC26-00NT40975

submitted to

**AAD Document Control, M/S 921-107
National Energy Technology Center
U.S. Department of Energy
P.O. Box 10940
Pittsburgh, PA 15236**

**Technical Monitor: Mr. Norman T. Holcombe
Contract Monitor: Ms. Crystal Sharp**

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

ABSTRACT

Application and testing of the new combustion Large Eddy Simulation (LES) code for the design of advanced gaseous combustion systems is described in this 12th quarterly report. In this quarter, continued validation and testing of the combustion LES code was performed for the DOE-SimVal combustor. Also, beta testing by six consortium members was performed for various burner and combustor configurations. A list of suggested code improvements by the beta testers was itemized.

Work will continue in FY04. A conditional modification to the contract will be granted. The additional work will focus on modeling/analyzing the SimVal experiments.

TABLE OF CONTENTS

	<u>Page</u>
Disclaimer	i
Abstract	ii
1. INTRODUCTION	1
2. EXECUTIVE SUMMARY	1
3. EXPERIMENTAL	1
4. RESULTS	1
4.1 DOE-SIMVAL Combustion Validation	1
4.2 Beta Test Results	3
4.3 Overall Results of the Beta Testing	9
APPENDIX A — Work Schedule	A-1
APPENDIX B — Future Plans	B-1
APPENDIX C — Solar Turbines' Final Report	C-1
APPENDIX D — Coen's Final Report	D-1
APPENDIX E — Virginia Tech's Final Report	E-1
APPENDIX F — University of California, Irvine (UCI) Final Report	F-1

LIST OF FIGURES

	<u>Page</u>
Figure 1. 3D LES Velocity Magnitude Snapshots of SimVal Experiment; $\phi=0.7$	2
Figure 2. 3D LES Temperature Snapshots of SimVal Experiment; $\phi=0.7$	3
Figure 3. Comparison of Measurements and Predictions at Premixed Exit Plane	4
Figure 4. Coen Case: Combustion Behind Duct Burner Element	5
Figure 5. CFD-ACE+ and FLUENT Comparisons; Use of LDKM Results in Less Dissipative Analysis	5
Figure 6. Sydney SMA2 Validation Case	6
Figure 7. Combustor Being Tested by Virginia Tech	7
Figure 8. Example of Temperature Snapshots	7
Figure 9. Schematic of BERL Furnace (Forniciari et al., 1994)	8
Figure 10. IFRF 300 kW Natural Gas Burner (Forniciari et al., 1994)	9

LIST OF TABLES

	<u>Page</u>
Table 1. List of Problems Reported by Beta Testers	10

1. INTRODUCTION

Vision 21 combustion systems will require innovative low emission designs and low development costs if Vision 21 goals are to be realized. In this three-year project, an advanced computational software tool will be developed for the design of low emission combustion systems required for Vision 21 clean energy plants. The combustion Large Eddy Simulation (LES) software will be able to accurately simulate the highly transient nature of gaseous-fueled turbulent combustion so that innovative concepts can be assessed and developed with fewer high-cost experimental tests. During the first year, the project included the development and implementation of improved chemistry (reduced GRI mechanism), subgrid turbulence (localized dynamic), and subgrid combustion-turbulence interaction (Linear Eddy) models into the CFD-ACE+ code. University expertise (Georgia Tech and UC Berkeley) was utilized to help develop and implement these advanced submodels into the unstructured, parallel CFD flow solver, CFD-ACE+. Efficient numerical algorithms that rely on *in situ* look-up tables or artificial neural networks were implemented for chemistry calculations. In the second year, the combustion LES software was evaluated and validated using experimental data from lab-scale and industrial test configurations. This code testing (i.e., alpha testing) was performed by CFD Research Corporation's engineers. During the third year, six industrial and academic partners are using the combustion LES code and exercising it on problems of their choice (i.e., beta testing). Final feedback and optimizations will then be implemented in the final release version of the combustion LES software that will be licensed to the general public.

2. EXECUTIVE SUMMARY

Work in this twelfth quarter (July - September 2003) has included further validation of the LES software. Predictions have been performed for the DOE-SimVal lean premixed combustor. Beta testing by gas turbine and industrial burner companies was completed. Itemized code improvements suggested by beta testers are in various stages of implementation into the code.

A conditional modification to the current contract will be granted for work to continue in fiscal year 2004. Details of the additional tasks to be performed are currently being negotiated.

3. EXPERIMENTAL

No experiments were performed this quarter.

4. RESULTS

4.1 DOE-SIMVAL Combustor Validation

In this quarter, CFDRC continued running the 3D LES case of the SimVal experiment. Sample snapshots of temperature and velocity are shown in Figures 1 and 2. These figures show the 3D flow structures that are present in the LES calculation. After approximately 6000 timesteps

(requiring approximately 3 weeks of run time), it was uncovered that the near wall turbulence (and heat transfer) was being calculated incorrectly for the LDKM subgrid turbulence model. As shown in the last quarterly report, the current calculation of heat loss in this combustor system is of paramount importance to accurately predict NO_x and CO emissions. Therefore, the 3D calculation is currently being repeated with near wall modifications. Results of the 3D calculation are not available at this writing.

Figure 1. 3D LES Velocity Magnitude Snapshots of SimVal Experiment; $\phi=0.7$

Figure 2. 3D LES Temperature Snapshots of SimVal Experiment; $\phi=0.7$

4.2 Beta Test Results

Solar Turbines: Solar Turbines ran a case that simulates fuel-air mixing in an industrial gas turbine pre-mixer. The pre-mixer geometry includes an upstream plenum that feeds a swirler. Fuel is injected through orifices located near the leading edge of the swirl vanes. The pre-mixer annular passage dumps into a combustor geometry. A 36 degree sector was modeled that included 1 swirl vane and fuel injected from just a single orifice (as in the experiment). Cyclic boundary conditions were needed for this configuration.

Solar desired to use a grid from Grid-Pro, another commercial grid generator. A substantial amount of time was spent by CFDRC to modify CFD-ACE+ so that Grid-Pro grids could be run (see previous quarterly report). In the end, we had to generate the grid with CFD-GEOM (CFDRC's grid generation package), and export the grid to Solar. However, Solar could not get CFD-ACE+ to run on their SGI computer system, so CFDRC had to set-up and run the case. The solution files were sent to Solar for post-processing.

Comparisons between measurements of fuel mass fraction and predictions are shown in Figure 3. Both RANS and LES predictions are shown in Figure 3. It can be seen that the LES predictions are in much better agreement with the measurements than the RANS predictions. It should be noted that four periodic slices are included in the prediction results in Figure 3 since the measurements consisted of a four-vane section. Peak concentration levels are well predicted, as well as the radial penetration.

A final report by Solar Turbines is included as Appendix C in this quarterly report.

Figure 3. Comparison of Measurements and Predictions at Premixed Exit Plane

Coen: Coen analyzed the combustion behind a duct burner element. Figure 4 presents the case analyzed. The fuel (CH_4) was injected uniformly from the downstream face of the bluffbody. Air enters from the right (Figure 4) at 10 m/s and 300K. Unfortunately, no data are available. Coen just compared CFD-ACE+ results with FLUENT results.

Coen used 2D LES analysis, utilizing approximately 10,000 cells. Subgrid models included LDKM for turbulence and assumed pdf (mixture reaction and progress) for turbulence-combustion interaction. The chemistry model employed was a one step-to-equilibrium products kinetics model with rates tuned for methane-air combustion. NO_x was post-processed.

Comparison of CFD-ACE+ and FLUENT predictions are shown in Figure 5. Since only snapshots are presented, it is impossible to assess differences between codes. However, the general flow features appear similar, except the CFD-ACE+ results appear less dissipative (i.e., finer structures are evident near the flameholder for CFD-ACE+). This is probably because the FLUENT predictions used the Smagorinsky model for turbulence, while the CFD-ACE+ predictions used the LDKM model (known to be less dissipative).

Coen commented that the grid generation was difficult to learn/understand because the technique/logic was completely different than FLUENT/Gambit. Other beta testers have stated just the opposite, indicating different users like different gridding techniques. A lot of time was spent in trying to install and use the Linux Operating System, resulting in less time available to assess the code. Coen commented that LES may not be ready for everyday CFD modeling when quick, straightforward answers are needed. However, Coen also stated that LES can (should) be used for selected cases that require accurate prediction of turbulence-combustion interaction.

Coen's final report is included as Appendix D in this quarterly report.

Figure 4. Coen Case: Combustion Behind Duct Burner Element

Figure 5. CFD-ACE+ and FLUENT Comparisons; Use of LDKM Results in Less Dissipative Analysis

Rolls Royce: Rolls Royce experienced a lot of difficulty installing the software on their computer system. This was especially true when Rolls-Royce went from Version 2002 (initial release of CFD-ACE+ at end of 3rd consortium meeting) to Version 2003 (released in Spring 2003). The main reason for the installation/running difficulty was that CFDRC did not support the Rolls-Royce computer system with Version 2003. Rolls-Royce also experienced difficulties when running on the PC cluster at the Pittsburgh Supercomputer Center (PSC), mainly due to downtime for security modifications and other issues. The PC cluster was also very slow (~400 MHertz chip speeds).

The case Rolls-Royce selected for code checkout/validation was the Sydney SMA2 Swirling Methane/Air Flame (see Figure 6). There were numerous issues that caused the case not-to-run/diverge on the PSC PC cluster. One major “bug” that caused significant problems (and took a long time to find) was the naming of the fuel as “FUEL”. This caused the restart file to be in error (mixture fraction was set to 0.0), and caused divergence in most cases when restart was invoked. When the 3D LES case was finally running, the flame appeared to be wrong (flame elongated). At this point, it was uncovered that pure methane was specified at the inlet, rather than methane and air at the inlet (user error). In the end, Rolls-Royce was not able to successfully run the case. However, a final report is in progress and lessons learned and suggested code improvements will be discussed.

Figure 6. Sydney SMA2 Validation Case

Virginia Tech: Virginia Tech has analyzed a premixed turbulent swirl stabilized combustor shown in Figure 7. Their desire is to use combustion LES to calculate the transfer function between u' and q' (for their reduced order models). Virginia Tech has performed high fidelity 2D LES calculations using LDKM, Chen’s mechanism, and LEM. An example of their

calculations is shown in Figure 8. There are no experimental data to date (but expected soon), so this should be considered a work-in-progress.

Virginia Tech's final report is included as Appendix E in this quarterly report.

Figure 7. Combustor Being Tested by Virginia Tech

Figure 8. Example of Temperature Snapshots

McDermott Technologies: McDermott continues to run the 300 kW BERL natural gas case (see Figures 9 and 10). A cold flow 3D LES case has been completed, and a hot 3D RANS case has been completed. The RANS simulation has gas temperatures that are 300-400K higher than measured. Radiation has recently been included in the RANS analysis, and temperature agreement appears to be better. The next step in the analysis process is to run 3D combustion LES analysis. We have not received a final report from McDermott Technologies at the writing of this quarterly, but expect to receive it in the next few weeks.

Figure 9. Schematic of BERL Furnace (Forniciari et al., 1994)

Figure 10. IFRF 300 kW Natural Gas Burner (Forniciari et al., 1994)

University of California, Irvine (UCI): UCI has significant problems/issues installing the software and running parallel simulations. All of these issues have now been resolved; however, little time was left to run meaningful LES simulations. CFDRC received UCI's final report, and it is included as Appendix F of this quarterly report. We expect UCI to continue using our software in the years ahead.

4.3 Overall Results of the Beta Testing

As we look at the results of beta testing, it is obvious that we underestimated the issues of software installation and parallel running on various computer platforms/operating systems. It took too much time to get the software up and running, leaving almost no time for the real task of running cases and validating the software against experimental data. Still, a significant number of "bugs" were identified and resolved. Table 1 presents a list of problems uncovered during beta testing. In addition, a number of code improvements were identified and suggested for implementation. Overall, the beta testing was very helpful in the software development-process, resulting in an improved software product. It also put the software in the hands of combustion system designers, and promoted the technology transfer of the software.

Table 1. List of Problems Reported by Beta Testers

1. Enthalpy was incorrectly computed using chem1 during startup. (2002-Keith)
2. SGS Kinetic energy computed incorrectly causing divergence. (All)
3. NEWRUN variable incorrectly set, causing startup problems. (2003-Keith)
4. Undefined boundaries internal to decomposed blocks after decomposition corrupts simulation setup. (Rolls-Royce)
5. Segmentation violation in input_ic_chem2. (Rolls-Royce)
6. DTF file locking causes deadlock on NFS mounted systems. (2002+2003-Babcox & Wilcox)
7. Vmix = 0.0. (2003-Keith)
8. Web help not working in release of V2003. (Coen Co, All)
9. Array not allocated; "indexing into array" error. (V2002-VT)
10. V2002 solver dies after specified interval output. (Babcox & Wilcox)
11. Multiple parallel bugs with cyclic boundaries. (V2002+V2003-Solar Turbines)
12. Unsupported Block Topologies for Plot3d in CFD-GEOM. (Solar Turbines)
13. SGI Origin 3000, and Mandrake Linux Cluster incompatible with our software. (Solar Turbines and Coen)
14. 2003 release of software excluded many operating systems previously compatible with 2002 code. (Rolls-Royce)
15. Many systems administered by industrial companies and organizations normally don't use "rsh" because of security reasons and prefer "ssh". (PSC, Virginia Tech, Coen, Babcox&Wilcox)
16. Incompatibility between 2002 and 2003 versions. (All)
17. Displaying of CFD-GUI and CFD-VIEW across the network almost never works because of software and/or hardware graphics. (All)
18. The parallel license wasn't setup correctly limited runs to serial and/or 4 processors initially. (UCI Irvine, Coen, VT, and Solar Turbines)
19. Like the ability to startup a parallel run on remote nodes excluding current login node. (Adrian Spencer, Babcox & Wilcox, VT)
20. Parallel setup difficult for normal users and requires expertise system administration. (All)
21. DTF key number must be repaired from Gridpro DTF output. (Solar Turbines)
22. 2002 Calls Error stop after checking J.Y. Chen reaction mechanisms. (naming convention problem)
23. 2002 and 2003 solver doesn't initialize fuel flow-field properly if "FUEL" is used for the fuel mixture name. (Rolls-Royce)
24. "Array Allocation Error". (Rolls-Royce)
25. Backflow in Outlet causing divergence. (VT)
26. Problems with slm_admin.exe. (UCI, RR)
27. dtf_decompose error for cyclic boundaries. (Solar Turbines, Babcox & Wilcox)

APPENDIX A — WORK SCHEDULE

Key Milestones

1 Complete In-Situ Adaptive Tabulation Module	7 Complete Tabulation Schemes
2 Complete LES Framework Modification to CFD-ACE+	8 Complete Parallelization of LES Code
3 Complete Reduced Mechanisms	9 Complete Implementation of LEM Model
4 Complete Selection of Cases	10 Complete Alpha Testing of LES Code
5 Complete Implementation of Turbulence Models	11 Complete Beta Testing of LES Code
6 Complete Implementation of Initial Version of LEM Model	12 Final Release of LES Code

Performance Targets

- A Alpha Release of LES Code
 B Beta Release of LES Code
 C Final Commercial Release of LES Code

- Planned
 Performed

APPENDIX B — FUTURE PLANS

The complete details of work to be performed in FY04 are being negotiated.

APPENDIX C

Solar Turbines’ Final Report

An Evaluation of CFDRC's LES Software For Design Of Combustion Systems (Beta Version)

**By
Alan J. Kubasco
Solar Turbines Incorporated**

1.0 INTRODUCTION

CFDRC is working to develop revolutionary Large Eddy Simulation (LES) CFD combustion simulation software to reduce development costs and time associated with industrial combustor development. The program focus is directed toward low emissions industrial combustors burning gaseous fuels. This work is funded under a DOE / Vision 21 contract.

A twenty-member consortium consisting of government, academic, and industrial organizations was formed to guide and direct software development. Selected industrial partners would act as beta testers to validate/compare predicted results against experimental data and to provide feedback to CFDRC for further software improvement.

This new software is being developed using Large Eddy Simulation (LES), which is better able to model the transient nature of gaseous-fueled turbulent combustion. Solar Turbines is particularly interested in the capabilities of this new software because it has the potential to accurately predict fuel-air mixing profiles. This is particularly important in the design of low emissions fuel injectors.

2.0 SOFTWARE EVALUATION/EXPERIENCE WITH SOFTWARE

Solar selected a Mars injector for which UC Irvine had previously acquired extensive velocity and fuel-air mixture fraction data for which to evaluate the LES software. The first step in this process was to generate the fluid

Solar® Turbines

A Caterpillar Company

mesh. Solar used the mesh generation software GridPro by Program Development Corp. of White Plains, NY. The CFD mesh of the Mars injector was completed in mid-February, 2003 and ready for incorporation into the LES software.

The next 3 months or so were spent trying to convert the GridPro mesh into the dtf format required by CFD-ACE -- we were unsuccessful. The conversion problem was complicated by the fact that the mesh was periodic. We will continue to work with the software developers from GridPro and CFDRC to solve this problem.

In order that Solar acquire some experience in using the LES software, the decision was made to have CFDRC mesh a model of a thick vane injector with a single fuel injection port in each vane for which Solar had fuel-air mixture data. CFDRC would setup the model in the LES mode, initiate the run, and then transfer the files to Solar for the completion of the run. Unfortunately, because of set-up and system incompatibility problems Solar has been unsuccessful in getting the LES software to run on our SGI system. We will continue to work with CFDRC and Program Development Corp. to solve both the grid conversion problem and the software installation issues.

CFDRC ran this model on a sixteen-node Linux cluster to 8000 sweeps. The data was sent to Solar for comparison to experimental data and will be discussed in a subsequent section.

Although we were not able to run the software, numerous exchanges between Solar and CFDRC called attention to a few shortcomings in the LES software. They are:

Boundary Selection: The boundary selection process seems to involve using the screen cursor to individually pick the boundary areas. This can be a tedious task on a complex geometry.

A competitor's software uses a screen cursor selected 'seed' vertex on one of the surface cells of the boundary to initiate the boundary surface selection. The selection process automatically continues until an edge is encountered. This is a very useful feature when working with intricate geometries that are not easily displayed in their entirety on the screen.

Model Decomposition: Although we have not had experience in setting up and decomposing a large model it is my impression that stopping a job and decomposing the model to run on a different number of nodes is not a simple process. In my experience with other CFD software this is a rather easy task, taking at most about five minutes for a large model.

Exporting Data To Third Party Post-Processors: The capability to export the data directly to a FieldView format for post-processing would be a significant benefit.

3.0 Experimental Setup

The experimental data used for model comparison was obtained at the exit plane of a ten-vane gaseous fuel injector. A photograph of a similar rig setup is shown below.

Solar® Turbines

A Caterpillar Company

The photograph shows the exit end of the injector with the fuel/air sampling rake in place. The sampling rake is comprised of a series of individual gas sampling tubes. The setup used to obtain the data to compare to the LES results used a six-probe rake system traversed over a 120-degree sector.

Each of the ten-swirler vanes used in the injector has six 0.052-inch diameter fuel injection ports located on the leading edge – three on the pressure side and three on the suction side. For this experiment five of the ports were blocked out using a metal foil tape leaving only one fuel injection port on the pressure side of the vane. The figure below shows the swirler and the location of the fuel port.

4.0 COMPARISON OF LES MODEL RESULTS TO EXPERIMENTAL DATA

Fuel-air mixture data from the experimental rig is shown in the contour plot below. The measurements indicate that mixing is more intense in the inner

Solar® Turbines

A Caterpillar Company

half of the annulus. One can still discern the pressure of the individual vanes and the single fuel injection ports in each vane. The counter-clockwise rotation of the flow and the rotation of the fuel concentrated zone can also be seen in the contour pattern.

FUEL-AIR MIXTURE DATA FROM EXPERIMENTAL RIG

(Fuel Mass Fraction – View Aft Looking Forward)

The LES model used the same boundary conditions as run in the experimental rig. They are as follows:

Solar® Turbines

A Caterpillar Company

Air Flow Rate = 0.3 lbm/sec

Air Temp = 540 R

Air Inlet Pressure = 15 psia

Fuel Flowrate = 0.002 lbm/sec

Fuel Temp = 540 R

Fuel Pressure ~15 psia

The LES prediction of fuel mass fraction at a plane corresponding to the fuel-air sampling rake is shown below. LES predicts a much higher concentration of fuel in air at the swirler hub.

LES FUEL-AIR MIXTURE PREDICTION
(Mass Fraction Fuel – View Aft Looking Forward)

Solar® Turbines

A Caterpillar Company

The pressure of the ten vanes - each with a single fuel injection port- are very evident. LES correctly predicts the counter-clockwise swirl component and the shearing action tending to mix and spread the fuel concentrated in the lobes.

Obviously, there is a difference between the LES prediction and the experimental data. This difference can be due to a number of factors associated with both the LES model and the experimental.

Possible Experimental Inaccuracies:

- 1) Metal foil tape was used to cover five of the six fuel ports on the vane leading edge. This tape is acting to some degree as a turbulence generator. How much additional mixing this is causing is unknown.
- 2) Six fuel-air sampling tubes were used with the inner and outer most tubes 0.045 inches from the walls. Therefore, the flow adjacent to the wall is not being sampled.
- 3) No attempt was made to sample the fuel-air mixture isokinetically.
- 4) The uniformity of the airflow into the swirler is an unknown.

Possible LES Modeling Inaccuracies:

- 1) Mesh density may not have been sufficient enough to resolve the finer details.
- 2) Uniform air velocity profile at the inlet may not replicate actual experimental conditions.
- 3) The fuel velocity profile may not reflect actual experimental conditions.

5.0 SUMMARY

The LES software predicts only the very general trend of the fuel-air mixing profile as measured experimentally. The high concentration of fuel in air predicted in the layer adjacent to the hub is unexpected and counterintuitive. Further investigation is needed to determine what in the LES software is causing this to occur.

APPENDIX D

Coen's Final Report

LES STUDY OF COMBUSTION IN DUCT BURNER

REPORT

Submitted To:
CFD Research Corporation
Huntsville, AL 35805

Prepared By:
Paul Matys
Coen Company, Inc.
Woodland, CA 95776
pmatys@coen.com

August, 2003

TABLE OF CONTENTS

Table of Contents.....	2
List of Figures.....	3
List of Tables.....	3
1. Introduction	4
2. Summary of Results.....	5
3. Comments	6
4. Figures	9

LIST OF FIGURES

Figure 1 Velocity Magnitude (m/s)	9
Figure 2 U-velocity component	9
Figure 3 Gas temperature	10
Figure 4 Mass fraction of methane.....	10
Figure 5 Mass fraction of CO	11
Figure 6 Mass fraction of oxygen	11
Figure 7 Mass fraction of CO2	12
Figure 8 Mass fraction of OH.....	12
Figure 9 Turbulent viscosity	13
Figure 10 Gas mixture density	13
Figure 11 Velocity Magnitude (m/s) - Fluent	14
Figure 12 U-velocity component - Fluent	14
Figure 13 Gas temperature - Fluent	15
Figure 14 Mass fraction of methane - Fluent.....	15
Figure 15 Mass fraction of CO - Fluent	16
Figure 16 Mass fraction of oxygen - Fluent.....	16
Figure 17 Mass fraction of CO2 - Fluent	17
Figure 18 Mass fraction of OH - Fluent	17
Figure 19 Turbulent viscosity - Fluent	18
Figure 20 Gas mixture density - Fluent	18

LIST OF TABLES

Table 1 Summary of inputs	4
---------------------------------	---

1. INTRODUCTION

The objective of this study was to predict unsteady gas flow and combustion behind a duct burner element using Large Eddy Simulation (LES). A commercial CFD code from CFD Research Corporation was evaluated. The results were compared with results from another commercial CFD package Fluent, and qualitatively with available measurements. The geometry of the duct burner was simplified to a bluff body with 2 baffles (see Figures 1), and was represented by 2D computational domain with about 10,000 computational cells. The air enters the duct at 10m/s and 300K. The fuel is released from the downstream face of the bluff-body at 0.5m/s and 300K. A summary of input data is shown in Table 1.

The CFDRC solver includes transport equations for mass, momentum, energy and species. The localized dynamic subgrid kinetic energy model (LDKM) was used to model turbulence. Combustion was modeled with PDF method and one step prescribed reaction. CO concentration was predicted by CO chemistry model, and thermal NO_x was calculated by the decoupled NO_x-model. For comparison, Fluent commercial code was used with very similar, however, understandably not identical physical models. For LES modeling Smagorinsky-Lily model was selected and for combustion PDF-model and equilibrium was used.

Table 1 Summary of inputs

Air inlet velocity	10m/s (33ft/s)
Air temperature	300K (80F)
CH4 inlet vel.	0.5m/s (1.6ft/s), 300K
CH4 temperature	300K (80F)

2. SUMMARY OF RESULTS

The results of the computer simulation are presented on a number of figures that show velocity field as contours of velocity magnitude and velocity components, gas temperature and species concentration.

Figures 1-10 show the results from CFDRC simulation. Velocity field downstream of the bluff body is characterized by a number of small flow structures as shown on Figure 1 and 2. The gas temperature is presented on Figure 3, there are number of high temperature zones resulting from combustion with the maximum gas temperature reaching slightly above 2200K. The concentration of species is shown on Figures 4-9. Methane quickly reacts into CO and hydrogen compounds. The mixing between oxygen and combustible gases in the bluff body wake seems to be limiting the overall fuel combustion rate. The flow field is fractionated into a number of recirculation zones of different scale.

The results from Fluent simulation are shown on Figures 11-20. The results are similar to CFDRC-results in qualitative sense. The gas temperature and concentration of species are roughly in the same range of values. There is, however, noticeable difference in the turbulent viscosity plots (Figures 9 and 19) with Fluent values being about 3 times smaller than those predicted by CFDRC software.

3. COMMENTS

- The initial difficulties at the early stage of the project were related to learning how to use CFDRC software. It was necessary to start with simpler cases and slowly progress to more complex cases that involved unsteady LES and eventually combustion. By far the most difficult and time consuming part of the case setup was to create geometry grid. The “GEOM” module of the software was found to be difficult to learn or understand as it is using technique/logic completely different from other commercial grid generation software (e.g. Fluent/Gambit).
- The parallel running of the software required the purchase of the necessary hardware and installation of Linux OS. Before this project the Linux OS has not been used on the company PC's, and there was very limited experience with this operating system at the administrative level. There were severe prolonged problems with the incompatibilities between the hardware and Linux version necessary to run CFDRC software. Red Hat OS could not be installed due to missing hardware support, and Mandrake Linux OS was not supported by CFDRC, and was giving warning and error messages. Substantial amount of time was devoted to troubleshoot this problem with some CFDRC help. In the end the newer version of Red Hat provided full hardware support. The other problems with parallel running on the Linux cluster was related to the administrative level of cluster software configuration. The permission between different cluster nodes were not working properly for a long time. In the end the parallel running capability on the Linux cluster was achieved, however, in the hindsight the time spent on the hardware software setup and initial troubleshooting was disproportionate as compared with the overall time assigned to this project.
- Initial Linux OS was selected for running and post processing of the results. However, eventually the case setup/pre processing and post processing was moved to Windows-PC due to insufficient support on Linux for other auxiliary software (word processing, graphics manipulation etc). The Linux OS system still seems to be reasonable choice for running the cases, specially due to easy

control from Windows-PC (e.g., using VNC, Exceed) and easy file transfer between Windows-PC and Linux-PC using Samba software.

- The newer version of CFDRC software suite V2003 is much improved in terms of help resources and user-friendliness, however, the difficult operation of GEOM module still seems to be the biggest stumbling block preventing from fast, efficient usage of the CFDRC-software.
- The test case for the LES combustion represents simplified geometry of the duct burner. In simplification the geometry consists of bluff body with baffles. The geometry can be treated as 2D. This assumption substantially reduces CPU-time required to run a case, on the other hand the 3D-effects of unsteady LES simulation are lost. In the future the results from 2D vs. fully 3D case will be compared and evaluated. At the present time, 2D case seems to be a good choice for the test case because it allows for quick running and troubleshooting of the cases, and quick learning of the fine tuning necessary to run unsteady LES-combustion cases.
- The initial results from CFDRC runs were suffering from unreasonably low combustion temperature. After some improvement in the grid and adjustments in the discretization schemes and under-relaxation factors, i.e., fine tuning of the model, the predicted temperature and flow pattern/structure were more reasonable.
- The initial runs with Fluent went smoothly, as there was previous experience with running unsteady LES-combustion with this software. The gas temperature was at reasonable level, and large structures (eddies) could be observed downstream of the bluff body. After comparing initial results from Fluent and CFDRC, it was concluded that large flow structures predicted by Fluent were not acceptable. The fine tuning of the Fluent model with the help of the Fluent-Support resulted in the quite dramatic change in the flow field with much smaller flow structures downstream of the bluff body, and good qualitative resemblance to the results obtained with the CFDRC software. This experience highlights the pitfalls of LES simulation which requires extensive testing and very careful selection of physical sub-models, discretization schemes, under-relaxation factors, time step and grid size.

- The experimental validation of unsteady LES combustion is very difficult, and at this stage for this project is limited only to qualitative comparison with the flow field observed in the test yard for similar geometries, and values of CO downstream of the flame. The prediction of CO by CFDRC or Fluent software varied widely between different combustion models. The final results from simulations were still almost by order of magnitude different from experimentally observed values for similar flow and combustion situation.
- At the present time, the unsteady LES combustion modeling seems to be in the early stage, both in terms of physical models and validation. More time is necessary to learn how to use the models and how to interpret the results. The LES simulations may not be ready yet to be used for everyday CFD modeling when quick and straightforward answers are expected from CFD modeling. However, even now, LES can be used for selected cases, where traditional modeling fails to give full and accurate prediction of the flow-combustion interaction. Even at this early stage of the software development the results from LES simulation provide valuable insight into the flow field and combustion process.

4. FIGURES

Figure 1 Velocity Magnitude (m/s)

Figure 2 U-velocity component

Figure 3 Gas temperature

Figure 4 Mass fraction of methane

Figure 5 Mass fraction of CO

Figure 6 Mass fraction of oxygen

Figure 7 Mass fraction of CO_2 Figure 8 Mass fraction of OH

Figure 9 Turbulent viscosity

Figure 10 Gas mixture density

Figure 11 Velocity Magnitude (m/s) - Fluent

Figure 12 U-velocity component - Fluent

Figure 13 Gas temperature - Fluent

Figure 14 Mass fraction of methane - Fluent

Figure 15 Mass fraction of CO - Fluent

Figure 16 Mass fraction of oxygen - Fluent

Figure 17 Mass fraction of CO₂ - Fluent

Figure 18 Mass fraction of OH - Fluent

Contours of Subgrid Turbulent Viscosity (kg/m-s) (Time=3.2155e+00)

Aug 21, 2003
FLUENT 6.1 (2d, segregated, pdf13, LES, unsteady)

Figure 19 Turbulent viscosity - Fluent

Contours of Density (kg/m3) (Time=3.2155e+00)

Aug 21, 2003
FLUENT 6.1 (2d, segregated, pdf13, LES, unsteady)

Figure 20 Gas mixture density - Fluent

APPENDIX E

Virginia Tech's Final Report

Simulation of Reacting Flows in a Premixed Turbulent Swirl Stabilized Combustor for Determination of Flame Dynamics

A Validation Study of the CFDRC LES Combustion Code

Progress Report Submitted to:

Dr. Steven M. Cannon
CFD Research Corporation
215 Wynn Drive
Huntsville, AL 35805

by:

Prateep Chatterjee – Research Scientist
Uri Vandsburger – Supervisor
Department of Mechanical Engineering, Virginia Tech
Blacksburg, VA 24061

September 5, 2003

Executive Summary

A CFD study of unsteady reacting flow process inside a combustor geometry has been performed to investigate the effects of acoustics on the unsteady heat release rate of a premixed gaseous flame. The motivation for undertaking the study is the problem of combustion instabilities which result from acoustic waves in the combustor perturbing heat release from the flame by generating fluctuations in flame surface area. These combustion instabilities are detrimental to the operation of gas turbine systems and must be controlled. Combustion control algorithms implement empirical or reduced order model based transfer functions between various variables, including the transfer function between unsteady velocity (u') and the unsteady heat release rate (q'). The objective of this study has been to calculate this transfer function. The transfer function calculated will in turn help in the development of reduced order models that will lead to elimination of combustion instabilities.

The CFDRC LES beta code has been used for calculation of the transfer function. Only part of the actual experimental combustor geometry (at Virginia Tech) has been used for the simulation to reduce the computation cost. Steady state inlet boundary conditions have been generated using experimental data and an unsteady component of velocity has been imposed at the inlet boundary. The resulting reacting flow field has been investigated and different unsteady phenomena have been seen to be occurring. Flame front instabilities have been seen to occur and data has been collected for subsequent investigations of the transfer function between u' and q' .

This document is only a progress report, since much work remains to be done. Among the issues to be addressed will be improving the accuracy of the imposed acoustic boundary conditions and the investigation of various unsteady phenomena observed in the present computations by conducting experiments and changing inflow conditions. Further validation of the LES code will be performed by collecting useful experimental data.

1 Introduction

Reduction of pollutant formation in gas turbines has been an important issue for gas turbine constructors. Lean premixed prevaporized (LPP) combustors are one of the choices, amongst several different methodologies being developed to match international emission standards. LPP combustors have reduced NO_x production but are susceptible to combustion instabilities [1]. Acoustic waves in the combustor perturb heat release by generating fluctuations in flame surface [2] and/or mixture fraction. If this unsteady heat release couples with acoustics, some eigenfrequencies of the combustor may be encouraged depending on the phase lag between acoustic waves and unsteady combustion. Understanding and preventing the resulting resonances are important issues in the development of LPP combustors.

Linear acoustics may be used to analyze and model combustion-acoustic interactions. These models are used to predict self-sustained frequencies in gas turbine combustors [3]. In these models, the flame is viewed as an acoustic device which generates an unsteady heat release depending on local acoustics. These simple interaction models usually comprise of a frequency dependent transfer function, relating heat release fluctuations with pressure and/or velocity fluctuations. The transfer functions (frequency response function – FRF) are not only dependent on frequency, but also depend on the combustor geometry, operation mode of the combustor and on the interaction of vortices with the flame front. It is possible to derive the FRFs using analytical calculations for simple combustor configurations, but the use of analytical models to obtain transfer functions for complex combustion systems have met with very limited success [4].

There is a need to develop reduced order models with the help of FRFs that describe the dynamic response of swirl stabilized flames to perturbations in the velocity and mixture fraction of the incoming reactants. These models are expected to be simple and yet, exhibit all the dominant dynamic characteristics of the combustion process.

Since a large number of physical variables are involved in the combustion process occurring in complex swirl stabilized combustors, simple systems (burner stabilized laminar flat flames) were initially studied here at Virginia Tech [5]. Having understood the dynamics of laminar flat flames and developed methodologies to build reduced order flame dynamic models, an experimental setup was designed and fabricated to study the dynamics of turbulent swirling flames at VACCG (Virginia Active Combustion Control Group) laboratory at Virginia Tech. The technique developed at VACCG to measure the open loop transfer function of laminar flat flame dynamics will be used for developing the transfer function for the flame dynamics (within the linear range) of swirl stabilized turbulent flames.

The experimental setup used to study the dynamics of turbulent swirl stabilized flames is schematically shown in Figure 1. The system consists of mass flow meters, an air-fuel mixing system, the flow control system, a turbulent variable swirl combustor, the dynamic velocity measurement system, the dynamic OH* measurement system, and the data acquisition system. The fuel flow measured using an array of mass flow meters is fed into a premixer that thoroughly premixes the fuel and air prior to the injection of the premixed charge into the combustor. Microphones were used to obtain dynamic velocity signal, while the OH* chemiluminescence captured by viewing the entire flame from the side was taken as the measure of the dynamic heat release rate. Controlled acoustic perturbations were imparted to the flow using a speaker. The dynamic signals were analyzed using the Hewlett Packard frequency analyzer, while the flow parameters were recorded using a data acquisition system.

The flame in the turbulent variable swirl combustor is stabilized by the presence of the central recirculation zone (CRZ) and the outer re-circulation zone (ORZ), as shown in Figure 2. The CRZ and the ORZ re-circulate the products of combustion back to the inlet of the combustor, thereby enabling the transfer of energy from the hot products of combustion to the incoming reactants. This fluid-dynamic feature of swirl stabilized flames that creates a continuous ignition source, eliminates the

Figure 1: Schematic of the experimental setup

need for an external energy re-circulator as was required for laminar flat flames. By altering the flow field and hence, the strength of the re-circulating zones, the flame could be forced to reside in either of the re-circulating zones or on the shear layer between the re-circulating zones. Such a variation in the flow field could be achieved by changing the swirl number and hence, the swirl strength of the flow entering the combustion chamber.

The variable swirl turbulent combustor was designed with a maximum pressure rating of 150 psig, a maximum thermal rating of 400 *kW* and is capable of accommodating 200 *SCFM* of total flow. It has a variable swirl generation arrangement that generates a maximum swirl number¹ (S_g), of 1.86.

Although OH* chemiluminescence is presently considered as the most accurate measure of the dynamic heat release rate, q' , its signal generates only qualitative results. The systems level analysis of the combustion process, however requires a quantitative result for q' . Thus, there is an urgent need to characterize the OH* signal and acquire capabilities to extract quantitative values of the dynamic heat release rate from the OH* signal. A study aimed at achieving the above goals would involve experiments on simple laminar combustors and physics based modeling or CFD analysis of the combustor. The comparison between the experimental and the computational results would then provide the calibration of the OH* signal.

The objective of this study is to perform a CFD analysis of the experimental combustor, in particular calculate the transfer function between u' and q' for the “flame dynamics” part of the closed loop dynamic system (shown in Figure 3). In this study,

¹Swirl number is defined usually in the following form: $S = \frac{\int_{R_i}^{R_o} \rho u_z u_\theta 2\pi r^2 dr}{\int_{R_i}^{R_o} \rho (u_z^2 - u_\theta^2 / 2) 2\pi R_o r dr}$ (refer Ribeiro et. al [6]), but in the experiments conducted at VACCG the swirl number calculated is the geometrical swirl number, defined as: $S_g = \frac{R_o \pi r_e}{A_t} \frac{(Tangential\ flow\ rate)^2}{(Total\ flow\ rate)^2}$, where R_o is the radius of the inlet of the quarl, r_e is the radius on which the tangential inlets are attached with respect to the center of the combustor and A_t is the total area of the tangential inlets.

the effect on q' because of equivalence ratio fluctuations, Φ' , has not been included. In the experimental combustor, Φ' is not present because the air and fuel are perfectly premixed.

Using the transfer function computed from the CFD results, a qualitative estimation of heat release can be obtained by comparing the CFD data with the experimental OH* qualitative data. Another objective of the study is to use CFD to explain near field acoustic effects on the heat release rate. These near field acoustic effects are at least two dimensional in nature (axi-symmetric) and are responsible for the excessive increase in the OH* chemiluminescence. They need to be accounted for in the systems level description of the combustion process, so as to accurately predict the occurrence of thermo-acoustic instabilities.

2 CFD Model

Computational studies of premixed combustion processes are often restricted by the limitations of the Reynolds average simulation (RANS) models [7] and the turbulent combustion models used to represent the mean flame and its effects on the flowfield. The concept of large eddy simulation (LES) thus provides a more natural approach to analyzing unsteady turbulent reacting flows. LES has the potential to provide quantitative data about flame response [8]. According to Kaufmann et al. [9], LES will provide reliable information for flame transfer functions in gas turbines only if all of the following requirements are matched:

1. Full 3-D compressible Navier-Stokes equations must be solved using structured/unstructured grids for complex combustor geometries,
2. Numerical methods with small dissipation/dispersion should be used,
3. LES models for flow dynamics in complex wall bounded flows are required,

Figure 2: Recirculation zones in the combustion chamber

Figure 3: System level description of the thermoacoustic combustion process

4. LES turbulence/chemistry interaction model must be valid for premixed flames,
5. Unsteady boundary conditions must be modified to inject controlled acoustic perturbations and force the flow without creating spurious modes.

Out of the five requirements listed above, Fluent [10] is able to satisfy the first requirement and to some degree the fifth requirement. Although dynamic subgrid scale LES models are available in Fluent, the lack of turbulence/chemistry interaction models prevents the use of the Fluent LES code in simulating unsteady reacting flows. Therefore, as an alternative to using the Fluent commercial code (which has been the code of choice for VACCG in past several years), the CFDRC LES beta code is being used to calculate the transfer function between u' and q' for reacting flow in the VACCG experimental combustor geometry.

The CFDRC LES code satisfies almost all the requirements cited above. Apart from matching the second requirement, which needs the implementation of third-order accurate finite element Taylor–Galerkin-type of schemes [11] or Chimera grid methods with overlapping domain interfaces, the CFDRC LES code is capable of producing flame transfer functions which will be accurate. Problems related to the first requirement are not important in this study because the experimental combustion chamber has a simple shape. As will be illustrated in the boundary conditions subsection, the last requirement is met to a high degree of accuracy, if not perfectly matched.

2.1 Computational Domain

The experimental setup internal geometry is shown in Figure 4. A long length upstream of the combustor in the setup was selected to eliminate any freestream turbulence. There are four distinct sections can be observed in the schematic: the axial air flow section (at the bottom), the straight section with the swirler, the convergent-divergent section and the combustor (dump) on top. Out of these four sections, the

combustor is the only section of interest for the CFD study. Therefore, only the top part of the experimental setup has been included in the CFD simulation. The LES computational domain is shown in Figure 5. The computational domain comprises of an annular inlet, a circular bluffbody which is aligned with the inlet plane, the diverging section (quarl), the bottom steel wall of the combustor, the quartz cylindrical wall and the outlet. The bluffbody diameter is 0.75" which is half the diameter of the inlet plane (1.5"). The inlet to the dump has a diameter of 2.5" and the outlet of the dump has a diameter of 4.92". The height of the combustor is 7.5" and this height has been selected so that the eigenmodes of the combustor do not get excited. Therefore, there are no self-sustained frequencies in the combustor. By eliminating self-sustaining frequencies and imposing velocity perturbations at the inlet (known as forcing), a transfer function between u' and the unsteady heat release rate q' can now be obtained.

2.2 Boundary Conditions

Eigenfrequencies strongly depend on the choice of acoustic boundary conditions. The resonant modes of a combustor depend on the acoustic boundary conditions at the inlet and outlet. The method used for inlet forcing should not affect these modes. The real combustor geometry (shown in Figure 4) has been simplified to perform an LES computation (inside the geometry shown in Figure 5) and inlet forcing is applied to the artificial computational inlet. The inlet forcing is a result of imposing an unsteady velocity which will make the computational inlet to act as a velocity node ($u' = 0$ at the velocity node) for waves reflected from the combustion chamber (the dump) to the inlet. The existence of this velocity node in the simulations (this velocity node is not present in the actual combustor) may perturb the results. While imposing velocity oscillations at the computational inlet, it has been assumed that because of the high acoustic impedance present at the flame location, any wave reflected from the outlet

Figure 4: Actual internal geometry of the experimental rig (the centerbody is shown with the dashed line)

Figure 5: LES computational domain

(which is a reflecting boundary with $p' = 0$) will be dampned by the flame front, in effect eliminating any outgoing waves at the computational inlet. Kaufmann et al. [9] have shown that unless outgoing waves at the computational inlet are eliminated from the forcing, accurate transfer functions can not be obtained. Since a large acoustic impedance is present at the flame front, we are assuming there are no outgoing waves present at the computational inlet.

The LES computation has been performed for an adiabatic case, which means all walls of the computational domain (including the bluffbody top) have been assumed to be insulated. The outlet is at atmospheric pressure (pressure node $\Rightarrow p' = 0$) and at the inlet a combination of mean and fluctuating velocities have been imposed. Since the computational inlet is way downstream of the swirler section of the actual geometry, mean flow velocity profiles at the computational inlet can not be assumed without knowledge of the mean flowfield in the actual combustor geometry. Therefore, hot-wire anemometry has been used to measure mean flow axial, radial and tangential velocities at the computational inlet plane location inside the experimental setup. Mean velocity profiles for a total flow rate of 20 *SCFM* and a geometric swirl number, S_G of 1.19 were recorded and are shown in Figures 6, 7 and 8.

Since the CFDRC code did not have an option for specifying a subgrid scale turbulent kinetic energy profile at the inlet, a mean value of $0.1 m^2/s^2$ obtained from the subgrid scale turbulent kinetic energy profile (shown in Figure 9) was specified. This assumption is not very accurate and most probably affects the shape of flame making it less compact. Also, instead of using the axial inlet velocity profile, a mean value of $7 m/s$ was specified. Sum of twenty sine waves with frequencies ranging between $25 Hz$ and $750 Hz$ have been included in the fluctuating velocity component. The following equation shows time dependent axial velocity specification at the computational inlet:

$$u = \bar{u} + u' = \bar{u} \{1 + 0.1 [\sin(2\pi f_1 t) + \sin(2\pi f_2 t) + \cdots + \sin(2\pi f_{20} t)]\} \quad (1)$$

Figure 6: u velocity profile at the computational inlet Figure 7: v velocity profile at the computational inlet

Figure 8: w velocity profile at the computational inlet

Figure 9: k_{sgs} profile at the computational inlet computed using u , v and w values

The fluctuating component (u') has been set to be 10% of the mean velocity ($\bar{u}$) and the frequencies f_1 to f_{20} lie between values of 25 Hz and 750 Hz . A premixed methane-air mixture of equivalence ratio, $\Phi = 0.75$ has been also imposed at the inlet. There are no fluctuations in equivalence ratio present in the computations.

2.3 Modeling Parameters

The LES computations have been done on a 2-D axi-symmetric grid, so as to reduce computation time. Full 3-D computations were planned originally, but have not yet been performed because of lack of computational resources. The 2-D axi-symmetric computations can be justified because the flame structure observed in the experimental setup is axi-symmetric. While performing the computations on the 2-D axi-symmetric domain, it was brought to our notice that during the alpha testing phase, 2-D axi-symmetric results showed inaccuracies. The problem was attributed to the lack of artificial viscosity in the LES code and velocity calculations near the axis were found to be inaccurate. We did not observe any such anomalies in our results and therefore, this approach was not abandoned in favor of 2-D/3-D computations.

2.3.1 Flowfield Modeling

The axi-symmetric solver was used with a time step size of $1 \times 10^{-4}\text{ s}$ and the Crank-Nicolson 2nd order accurate time integration scheme was applied. The time step size was chosen based on the Kolmogorov time-scale $(\nu/\epsilon)^{1/2}$. The Kolmogorov time scale was estimated by running a steady state RANS case using a single-step chemistry model. From the RANS results, the R grid factor was also calculated and was found to be less than 1 for almost all of the computational domain, except at the inlet section where small patches of higher values of the grid factor were observed to occur. These patches (with $R \leq O(10)$) were seen to occur randomly over the quarl inlet section of

the domain, however running the RANS solver for several thousand iterations resulted in fewer occurrence of the patches.

In the flow modeling panels of the code, the swirl option was chosen and the reference pressure was kept at $101,325 \text{ N/m}^2$. Six iterations per time step with a convergence criteria of 10^{-4} kept the computation time per time step very low (of the order of 12 s on a dual Athlon 2 GHz machine with 2 GB RAM). The 2nd order limiter spatial discretization scheme was used for velocity, turbulence, enthalpy and species whereas central differencing was used for density spatial discretization. A blending factor of 0.1 was used for each of the discretization schemes. The AMG solver has been used for each of the variables. Density was modeled as an ideal-gas, viscosity using the mixture Sutherland's law, specific heat using the mixture JANNAF method, thermal conductivity by specifying the Prandtl number to be 0.707 and mass diffusion modeled by specifying the Schmidt number to be 0.7. The localized dynamic kinetic energy model (LDKM) was used to model the sugrid scale stresses. This model was preferred over the Smagorinsky model because even though LDKM is equally computationally expensive, it provides better estimation of the subgrid stresses as compared to the Smagorinsky model.

The solution was initialized by assuming air-fuel mixture to be present in the quarl section and air to be present in the downstream dump section. High temperature was patched in the immediate downstream of the quarl section for ignition.

2.3.2 Chemistry Modeling

Although curve fit mechanisms are known to produce good estimations of the laminar flame speed, for dynamic modeling of the flame (especially while finding the flame response function, FRF) they are not able to capture the flame dynamics accurately. Therefore, reduced reaction models which have been tuned for modeling premixed flames need to be used. The 19 species 15 reactions methane-air model by J. Y.

Chen which is built into the CFDRC code, has been chosen to model chemistry in the CFD analysis. This model, apart from being tested in other validation cases for premixed flame modeling, also includes radicals and minor species like OH and $HCHO$, which are useful in making an estimate of the heat release rate from the combustion process and are particularly useful for validation against experimentally obtained PLIF results.

The turbulence-chemistry interaction modeling is one of the most important aspects for modeling acoustic-flame interactions. Therefore, the subgrid linear eddy model (LEM) has been chosen based on a literature survey which proved LEM's capability of modeling turbulence-chemistry interactions in unsteady reacting flows. It was known prior to the beginning of the beta testing that LEM will prove to be computationally expensive, but the decision of opting for 2-D axi-symmetric modeling eased some of the computational restriction associated with the model. Also, since there were no benchmarks available in literature for the mixture fraction/PDF method in unsteady acoustic-flame interaction modeling, LEM seemed to be the only option left (there is considerable amount of literature available for "dynamically thickened flame model" and the BML model applications in modeling acoustic-flame interactions though).

3 Results and Discussion

The results section has been divided up into two subsections. In the first section, the reacting flowfield structure has been discussed and time averaged contour plots of velocities, temperature and species have been shown. The second subsection showcases the response of the flame to the imposed unsteady velocity perturbations. All results included in this section are for a flow rate of 20 *SCFM*, geometric swirl number (S_g) of 1.19 and $\Phi = 0.75$.

3.1 Reacting Flowfield Structure

Real time data obtained from the LES code was analyzed to obtain time-averaged profiles of velocities, temperature and species mass fractions. Time averaging was carried out for 2 s of data (with a sampling rate of 10,000) for the flow variables. Time averaged velocities and temperature contours are shown in Figure 10. From the figure, it can be noted that the overall structure of the flame has been captured, including the inner and outer recirculation zones (as can be seen in the u velocity contours plot). As expected, the temperature at the inner recirculation zone is high compared to the outer recirculation zone. This high temperature recirculating fluid in the inner recirculation zone is responsible for the constant ignition of the fresh incoming air-fuel mixture. It should be noted that the time averaged v velocity shows an irregular contour pattern inside the quarl, indicating an unsteady process happening at that location. The swirl component of velocity (w) shows a decay from the inlet to the mid-combustor region. Swirl is the highest inside the quarl.

Figure 11 shows the contours of time averaged CH_4 , O_2 , CO_2 and H_2O mass fractions. The CH_4 mass fraction contours indicate that all the fuel is burnt just downstream of the combustor inlet plane. The actual swirl number (S) is approximately 0.27 and therefore the flame shows less compactness compared to higher swirl number flows. The contours of O_2 , CO_2 and H_2O indicate some recirculation at the outlet plane. The recirculation observed in the data has not been validated against experimental observations.

An unsteady phenomena has been observed to occur in the combustion process. Figures 12, 13, 14 and 15 indicate the presence of vortex shedding inside the quarl section. Each of the figures include 10 plots of instantaneous contours of u , v , w velocities and temperature. Each plot corresponds to $\pi/5$ angle for one cycle of the vortex shedding (time difference between each plot is 4×10^{-3} s). By performing a spectral analysis, the vortex shedding frequency was estimated to be approximately 255 Hz. The vor-

Figure 10: Contours of time averaged u , v , w (m/s) and temperature (K)

Figure 11: Contours of time averaged mass fractions of CH_4 , O_2 , CO_2 and H_2O

tex shedding phenomena is clearly visible in Figure 13 which shows the contours of v velocity. The swirl velocity also indicates the presence of vortex shedding, although the radial velocity is a much more clear indicator. The temperature contours in Figure 15 show that the flame shape itself is changing as the vortex passes through the flame every $4 \times 10^{-3} s$. This vortex shedding phenomena was not observed in the experimental studies, although the flame surface exhibited a $275 Hz$ azimuthal mode as seen in phase-locked CCD camera images of the flame. Basically, three alternate explanations can be given to explain this phenomena:

Acoustic waves: Both the inlet and outlet are acoustically reflecting boundaries and therefore, the unsteady phenomena which seems to be vortex shedding can actually be a wave which is getting reflected at each boundary and is creating an unsteady response from the flame while passing through it.

Vortex-flame interaction: The vortex shedding is observed to happen at the quarl walls, which is an indicator of a shear layer instability. This kind of instability is seen to occur in diffusers. In the experimental setup, the vortex shedding was not seen to occur, whereas the CFD results show the $255 Hz$ phenomena. A possible reason for this anomaly can be attributed to the inaccurate k_{sgs} profile specification at the inlet as well as inaccuracies in the inlet boundary condition velocity profiles. The hot-wire data collection at the narrow $9.5''$ inlet plane annular duct only yielded 6 or 7 points and velocity data for these few points were then used to create the profiles. Slight inaccuracies in specifying the velocity profiles and the turbulence inlet conditions can lead to such shear layer instabilities to occur.

Flame response to azimuthal acoustic wave: Phase locked images of a $275 Hz$ flame flapping phenomena were recorded using a CCD camera. The flame was excited by a $275 Hz$ acoustic wave and CCD camera images were taken by phase-locking the camera with the acoustic-driver unit. While taking phase

locked images for a 100 Hz excitation frequency, this “flapping” phenomena was not seen to occur. It is possible that the 275 Hz frequency was always present independent of the frequency of excitation. Phase locking at 100 Hz effectively eliminates any higher frequency content from the images, which means that any higher frequency content (in this case the 275 Hz frequency) was not observed for the 100 Hz excitation case.

A final explanation of the 255 Hz phenomena observed in the CFD results can only be given after closely investigating each of the three possibilities listed above.

3.2 Excited Flame Response

Once a frozen flowfield was achieved, flow forcing at the computational inlet was imposed. Figure 16 shows the time trace of unsteady inlet mass flow rate. As can be seen from the figure, multiple frequency content in the signal are present. Since we are studying linear response of the flame to incoming mass flow rate oscillations, we can safely impose a complex mass flow signal at the inlet (our mass flow rate signal contains 20 discrete frequencies). The power spectral density (PSD) plot of the mass flow oscillations at the inlet can be seen in Figure 17. Each of the twenty frequencies are visible in the power spectrum and each one of them has the same PSD magnitude.

The imposed mass flow rate oscillations result in an unsteady response from the flame. The unsteady temperature measured at different locations in the computational domain show different levels of excitation that results from the imposed mass flow rate oscillations. Figure 18 shows the PSD of temperature at six different locations inside the combustor. The outer recirculation zone temperature does not show frequency content beyond 100 Hz . Since the outer recirculation zone acts like a dampner for high frequencies, the PSD does indicate a correct trend. The inner recirculation zone and the mid-combustor $1/2$ diameter measurement points lie in a straight line.

Figure 12: u velocity contours for one cycle of vortex shedding (255 Hz). Each contour plot corresponds to $\pi/5$ radians increment

Figure 13: v velocity contours for one cycle of vortex shedding (255 Hz). Each contour plot corresponds to $\pi/5$ radians increment

Figure 14: w velocity contours for one cycle of vortex shedding (255 Hz). Each contour plot corresponds to $\pi/5$ radians increment

Figure 15: Temperature contours for one cycle of vortex shedding (255 Hz). Each contour plot corresponds to $\pi/5$ radians increment

Figure 16: Time trace of inlet normalized mass flow fluctuations

Figure 17: Power spectral density of fluctuating component of inlet mass flow rate

Therefore, it is not surprising to similar levels of PSD for each curve. Sharp peaks can be seen for the inner recirculation zone curve. These peaks are not present in the mid-combustor 1/2 diameter location because the frequencies get damped as we move downstream from the inner recirculation zone. The quarl location and the combustor inlet location both show distinct peaks for each of the frequencies of excitation. The 255 Hz peak is also clearly visible for the two curves. This peak is not visible in the mid-combustor 1/4 diameter location because of the downstream measurement location.

The main objective of this study has been the investigation of unsteady flame response due to an imposed upstream velocity perturbation. The frequency response function (FRF) or the transfer function needs to be obtained between unsteady heat release rate and unsteady velocity. This exercise has not been completed as yet because of the unexplained phenomena observed in the results. As a first estimate of the FRF, resulting temperature oscillations has been taken as the output and the inlet mass flow rate oscillations has been considered to be the input. The FRF magnitude is shown in Figure 19. The magnitude prediction is seen to follow a trend and a 40 dB drop between 25 Hz and 750 Hz can be observed. The phase relationship between u' and T' has not been predicted correctly by the CFD model. All efforts are presently being directed toward understanding the unsteady heat release phenomena and correct phase predictions between u' and q' needs to be obtained.

4 Ongoing and Future Work

The reported data is part of an ongoing investigation at VACCG in applying CFD to create transfer functions which are to be used in reduced order modeling of the unsteady combustion phenomena. Therefore, investigation into various phenomena influencing the unsteady flame response is being continued. Presently, data is being

Figure 18: Power spectral density of fluctuating component of temperature at different locations inside the computational domain. The mid-combustor location (1/2 diameter) is 3.75" downstream of the inlet plane of the combustor and lies on the centerline, whereas the corresponding mid-combustor (1/4 diameter) location is radially at a distance of 1.23" from the centerline

Figure 19: FRF between u' and T' : magnitude shown for two locations. The mid-combustor locations were selected because these two locations are downstream of the flame and are not affected by the near-field effects such as the 255 Hz phenomena

collected for the swirl combustor using the CFDRC LES code and FRF between area integrated heat release rate and velocity perturbation will be calculated subsequently.

The following issues will be addressed in the coming months:

1. Non-reflecting inlet boundary condition will be implemented. This implementation may lead us to produce accurate FRFs (particularly the phase information between u' and q').
2. Comparisons between unsteady temperature measurements inside the experimental combustor and predicted values from the CFD results will be undertaken (experimental studies currently ongoing).
3. Investigation into the 255 Hz phenomena is of high priority. Changes in the inlet boundary condition (including increasing the swirl number) will be undertaken.
4. A change in diffuser angle and the resulting effect on the vortex-shedding phenomena inside the quarl will be investigated.
5. The inlet boundary location and the effect of the downstream conditions on the inlet boundary condition will be investigated.

References

- [1] G. J. Bloxsidge, A. P. Dowling, and P. J. Langhorne. Reheat buzz: An acoustically coupled combustion instability. part 2. theory. *Journal of Fluid Mechanics*, 193:445–473, 1988.
- [2] A. P. Dowling. Nonlinear self-excited oscillations of a ducted flame. *Journal of Fluid Mechanics*, 346:271–290, 1997.
- [3] K. McManus, T. Poinsot, and S. Candel. A review of active control of combustion instabilities. *Progress in Energy and Combustion Science*, 19:1–29, 1993.
- [4] M. Fleifil, A. M. Annaswamy, Z. Ghoniem, and A. F. Ghoniem. Response of a laminar premixed flame to flow oscillations: A kinematic model and thermoacoustic instability result. *Combustion and Flame*, 106:487–510, 1996.
- [5] P. Chatterjee. Computational fluid dynamics studies of premixed laminar and turbulent combustion dynamics. PhD Dissertation Proposal, 2002.
- [6] M. M. Ribeiro and J. H. Whitelaw. Coaxial jets with and without swirl. *Journal of Fluid Mechanics*, 96:769–795, 1980.
- [7] C. G. Speziale. *Simulation and Modeling of Turbulent Flows*, chapter Modeling of Turbulent Transport Equations. Oxford University Press, New York, 1996.
- [8] T. Poinsot and D. Veynante. *Theoretical and Numerical Combustion*. RT Edwards, Philadelphia, 2001 edition, 2001.
- [9] A. Kaufmann, F. Nicoud, and T. Poinsot. Flow forcing techniques for numerical simulation of combustion instabilities. *Combustion and Flame*, 131:371–385, 2002.
- [10] Fluent Inc. *Lebanon, NH*.

- [11] T. Schonfeld and M. Rudgyard. Steady and unsteady flow simulations using the hybrid flow solver AVBP. *AIAA Journal*, 37(11):1378–1385, 1999.

APPENDIX F

University of California, Irvine (UCI) Final Report

Technical Brief

**Implementation and Application of Large Eddy Simulation of an
Axisymmetric Confined Flowfield In the Presence of a Bluff Body**

Q.Wang, T. Yoshimura, V. Jermakian, and V.G. McDonell

University of California
Irvine, CA 92697-3550
www.ucicl.uci.edu

UCI ARTR 2003-12

15 July 2003

Contents

List of Figures.....	ii
List of Tables.....	ii
Executive Summary.....	1
1 Introduction.....	2
2 Problem Statement.....	2
3 Validation Approach	2
4 Experimental and Numerical Systems	2
4.1 Test Stand.....	2
4.2 Test Hardware Geometry and Operating Conditions	4
4.3 CFD Model Information	5
4.3.1 Grid	5
4.3.2 LES Modeling.....	5
4.4 Digital Particle Image Velocimetry.....	5
4.5 Simulation Platform	7
5 Results.....	9
5.1 Software Implementation Experience	9
5.1.1 Installation	9
5.1.2 Parallel Run.....	9
5.1.3 Run CFDACE on X terminal	9
5.1.4 Time Limit.....	10
5.2 Measured Flowfield	10
5.3 Predicted Flowfield.....	11
5.4 Comparison Between Predicted and Measured Flowfield.....	13
5.4.1 Time Averaged V Velocity	13
5.4.2 Time Resolved Axial Velocity	14
6 Conclusions and Recommendations	16
6.1 Conclusions	16
6.2 Recommendation	16
7 References.....	16
Appendix	16

List of Figures

Figure 1. Rich Burn Quick Mix Lean Burn Combustion System.	2
Figure 2. Experimental Facility.....	3
Figure 3. Details of Test Section.....	3
Figure 4. CAD Drawing of the test hardware setup for LES and DPIV.	4
Figure 5. Details of geometry utilized for CFD setup, dimensions are in inches.	4
Figure 6. Structured Grid for Test Case.	5
Figure 7. Digital Particle Image Velocimetry Diagnostic experimental setup.	6
Figure 8. Particle Image in the combustor.....	7
Figure 9. UC Irvine Beowulf Cluster.....	8
Figure 10. PIV Diagnosis Region in the combustor.....	10
Figure 11. Time averaged Flowfield.....	11
Figure 12. Instantaneous Flowfield	11
Figure 13. RANS Flow Field (Velocity Magnitude)	12
Figure 14. LES Average Flow Field (Velocity Magnitude)	12
Figure 15. Time Averaged V velocity	13
Figure 16. Comparison of RANS and DPIV Axial Velocities.....	14

List of Tables

Table 1. Summary of Operating Conditions.	5
--	---

Executive Summary

This technical brief documents the implementation and application of a Large Eddy Simulation package provided to the UCI Combustion Laboratory (UCICL) by CFD Research Corporation (CFDRC). The software was obtained in early January 2003 and installed on the MPC (Medium Performance Computing) Beowulf Cluster located in University of California, Irvine. This cluster consists of 16 full-time nodes (32 CPUs) and approximately 150 part-time PC lab nodes. More information on this Beowulf Cluster can be obtained at <http://mpc.uci.edu/>. The test case identified for application of the LES software involved an axisymmetric, confined flow interacting with a bluff body. This test case is associated with a project examining the interaction of jets with a rich product stream at elevated temperature and pressure. The specific test section considered in the present effort is that associated with the rich product generation system. As part of the development of the rich burn section, better understanding of the flowfield generated by the section was desired.

In parallel with the LES simulations, experimental methods were also applied. In the present study, digital particle image velocimetry (DPIV) was utilized. The DPIV system provided instantaneous flowfield maps that could be compared directly the LES simulations. The flowfield was not “driven” by external means. As a result, the ability to specifically “synchronize” time variant behavior between the simulations and measurements was not possible. However, general trends in terms of the instantaneous and time averaged behavior were compared.

The installation process involved more time than anticipated, especially the implementation of parallel operation. Some of the reasons for the additional time included licensing installation troubleshooting and software interaction with Portable Batch System. In addition, challenges were realized due to the manner in which the parallel system use was managed. Specifically, limited use and scheduling of the parallel resources led to automated job termination prior to satisfactory convergence, which caused significant inefficiency in running the case. Attempts to overcome these handicaps were partially successful, but won't be fully realized until the future.

The results indicate that reasonable agreement between RANS and measurements was achieved. The comparison of time averaged LES solutions and time averaged DPIV results indicates similar agreement, but the total time available for the LES solution was insufficient to make detailed conclusion. The direct quantitative comparison of instantaneous behavior is difficult in the present case, because of limited total solution time for the LES and because the flowfield was not regularly periodic in time, but the general trends in terms of global flow field motion documented by DPIV were reflected in the LES results.

1 Introduction

In December of 2002, UC Irvine was provided with a beta version of the CFDRC Large Eddy Simulation code V2002.2.24 and asked to evaluate the utility and performance of the software by applying it to a problem of interest.

2 Problem Statement

The device selected to apply the LES software to is shown in Figure 1. The development of the rich-burn portion of a Rich-burn, quick-mix, lean-burn combustor is of direct interest. The pre-mixed section leads into the rich-burn section with a bluff-body flame holder. The bluff body is designed to create a re-circulation zone that can confine the primary reaction zone aerodynamically. The bluff body is also designed to prevent flashback by creating a pressure drop, which in turn develops flow speeds exceeding any flame speed. The flame holding section is the basis for the modeling and the experiment documented in this brief. Additional details regarding the geometry of the test device are provided below.

Figure 1. Rich Burn Quick Mix Lean Burn Combustion System.

3 Validation Approach

The approach selected for validation of the LES simulation involved measurements obtained using planar measurements and visualization. Because the LES simulation provides time resolved information about the full flowfield, diagnostics were selected to provide comparable information. Specifically, digital particle image velocimetry (DPIV) was utilized to provide instantaneous planar velocity measurements.

Several experimental conditions were selected for measurement and simulation. These cases were run in parallel and ultimately compared both qualitatively and quantitatively.

4 Experimental and Numerical Systems

4.1 Test Stand

An existing atmospheric pressure test stand was utilized for the study. The bluff body and fuel pre-mixing sections were mounted on to the test stand in an upfired orientation. The rest of the combustor was replaced with a quartz liner of 3.15" with a length of 12" in order to provide optical access for DPIV measurements. In addition to the quartz liner, optical flats were installed on the laser sheet entry and exit portions of the liner to minimize reflections and to improve the quality of the images. A photograph of the overall experimental setup is shown in Figure 2.

The air mass flow rate was metered with a critical flow orifice (0.18" dia.) and established by monitoring the pressure upstream of the orifice. The critical flow orifice flow rate vs. upstream pressure was established by means of a NIST traceable laminar flow element (Meriam LFE). The pressure drop across the bluff body was monitored using a water manometer.

Figure 2. Experimental Facility.

Additional details of the test section setup are illustrated in Figure 3.

Figure 3. Details of Test Section.

4.2 Test Hardware Geometry and Operating Conditions

The test hardware only receives non-preheated air and flows from the seeding system. The air is injected with a 1" tube that is connected to the air supply. The seed flows are injected through a ½" "T" fitting. The details of the seeding system are described in Section 4.4. An expansion section increases the area of the air and seeder flow by transitioning from a 1" tube to a 3.15" tube. After the expansion section a Vortab® inline flow conditioner is used to provide a uniform concentration and velocity before the bluff body inlet plane. The bluff body section consists of a 45° cone with a 2.99" diameter at the throat. The bluff body creates the re-circulation zone desired for flame holding in a combustor. The strength of the re-circulation zone depends on the air mass flow rate and associated pressure drop across the bluff body. The stagnation point in the re-circulation was assumed to be 3" from the face of the bluff body. The 8" quartz section was an adequate length for testing purposes and prevented ambient air from entering the flow path. The details of the test hardware geometry are shown in Figure 4. Note that the apparent opening in the bluffbody shown in Figure 4 (utilized for the ignitor assembly) was actually filled with a solid tube so that no air flow could enter this passage.

Figure 4. CAD Drawing of the test hardware setup for LES and DPIV.

The CFD geometry and grid are based on the simplified geometry illustrated in Figure 5. The bluff body geometry and the flow area are the most critical regions for the non-reacting flowfield. Preliminary CFD cases were run during the design process of the bluff body in order to establish a suitable pressure drop and recirculation zone for the flows of interest to the project.

Figure 5. Details of geometry utilized for CFD setup, dimensions are in inches.

The operating condition for the DPIV experiment is summarized in Table 1 and was utilized as the inlet conditions for the CFD simulations.

Table 1. Summary of Operating Conditions.

Orifice Pressure (Pa)	Air Flow Rate (kg/s)	Pressure Drop (Pa)	Temperature (K)
153,063	0.01386	600	300

4.3 CFD Model Information

4.3.1 Grid

The grid developed for the case features and ~1.0 Million cell structured grid, as shown in Figure 6. This grid is a 3D asymmetric grid and was refined several times until the best convergence was achieved.

Figure 6. Structured Grid for Test Case.

4.3.2 LES Modeling

The LES model implemented utilizes the following parameters and boundary conditions:

- $\delta t = 2.5 \times 10^{-4}$ s
- LDKM Subgrid Models
- Default Model Constants
- User Output 100HZ
- Fixed Velocity Inlet
- Fixed Pressure Outlet
- Random Fluctuations

4.4 Digital Particle Image Velocimetry

Digital Particle Image Velocimetry (DPIV) is a non-intrusive whole-flow-field technique providing instantaneous velocity vector measurements in a cross-

section of a flow. By using a 2D DPIV system two velocity components are measured, in the present case axial and radial components in the axisymmetric setup. The use of modern CCD cameras and dedicated computing hardware, provides the opportunity to measure velocity fields in essentially real time.

Details of the equipment utilized for this specific project include:

- Nd:YAG Laser : Continuum Surelite III@532nm, 340mJ/shot
- CCD Camera : Kodak Model ES 1.0
- Pulse Generator : Stanford Research Systems DG535
- PIV Software : TSI Insight ver.3.34

The laser, camera, and the associated sheet forming optics are illustrated in Figure 2. A schematic of the overall system is shown in Figure 7.

Figure 7. Digital Particle Image Velocimetry Diagnostic experimental setup.

A key requirement for DPIV is the presence of scattering centers in the flow. For the present study, nominally 2.0 micron alumina particles (MicroAbrasives GB1500) were utilized. They were introduced into the air stream approximately 500 mm upstream of the bluff body face using a slurry nebulization approach which is described in detail in Ikioka, Brum, and Samuelsen (1983). This method provides sufficient particle concentration and steady generation, key requirements for high quality DPIV results.

In DPIV, velocity vectors are calculated from the displacement of particles between two laser light pulses. The particles in the flow are illuminated in the diagnosis region with a laser light sheet. When the particle is illuminated, the scattered light image is captured by CCD camera. Through a technique called “frame straddling”, the Kodak CCD camera is able to obtain a second independent image from the scattered light from the second laser pulse. With the two separate images of the light scattered from the two consecutive laser pulses, the PIV software then establishes the displacement of particles using cross correlation methods, and through calibration of pixel spacing to physical distance, determines the velocity maps.

A cylindrical quartz is used for the flow measurement in the combustor and makes the flow in the combustor visible. However, light scattering from the surface of the quartz creates a challenge in terms of noise. This noise causes some errors in the vector field when the intensity of the surface scattering is comparable to the intensity of the light scattered by the particles. The software cannot discriminate between surface reflections and particle scattering. As a result, non-gloss black paper was carefully positioned on one side of quartz to mask any surface reflections from entering the camera.

A typical single image is shown in Figure 8. The size of this image is 107.9mm(H)*109.0mm(V). The scattering from the quartz on either the right or left edge of the particle field has been eliminated through the spatial filters and the scattering has been removed from the region of interest (i.e., the particle field). The bright part of the bottom is the flange around the bluff body which is the inlet plane. A substantial number of particles in the region of interest are noted as required for accurate image analysis.

Figure 8. Typical Particle Image in the combustor.

4.5 Simulation Platform

Figure 9 illustrates the MPC Beowulf Cluster at UCI, which was utilized as the simulation hardware platform. The MPC (Massively Parallel Computing) Beowulf Cluster is available to all UCI Researchers and it consists of 16 full-time nodes and approximately 150 part-time PC Lab nodes. The 16 full-time nodes are dual-CPU Xeon 2.0GHz and Athlon 1.8GHz units, which are completely dedicated to the cluster. The other 150 part-time nodes are lab PCs located in different PC labs on UCI campus, which are used by UCI students in daytime and as part of the computing cluster at night (10:00PM-7:00AM) when they are free of use.

The MPC was continually upgraded throughout the duration of this project, and plans remaining include adding 16 dual CPU compute nodes and a GigE switch will be added to MPC. The 16 dual CPU nodes will use the new AMD 64 Bit Opteron nodes and four of them were put into use in early May and were utilized by this project. Each of the 16 additional nodes will have dual Opteron Model 240 - 1.4GHz CPU / 1MB L2 Cache, 1GB PC2700 DDR main memory, Broadcom NIC GIGE * 2, internal disk, CD and floppy. However, since the MPC cluster is not currently running the 64-bit Redhat9 linux OS, the Opteron nodes are restricted to 32-bit performance.

The operation system of MPC cluster is RedHat Linux9.0. At the UCICL offices, Exceed 7.0 X terminal simulator was utilized on a Windows XP machine since the experience with Linux was minimal. Xterm was typically used to launch tasks on the cluster.

The MPC Beowulf cluster uses PBS (Portable Batch System) to manage its task queue. All the tasks must be submitted to the head node and appointed to the computing nodes by the head node. The head node does the maintenance of the queue and balance the load of the different computing nodes. This mechanism is good for optimizing the usage of the computing capability of the whole cluster. In order to prevent individuals from claiming all the computing resource of the cluster for a unreasonably long time, a 24 hour time limit was applied to each task submitted to the PBS.

For the current LES project, the four new dual CPU Opteron nodes were utilized. The task was split into 8 parts and carried out by parallel runs within 24 hours. The software package we used was CFDACE+ Ver2002.2.24.

Figure 9. UC Irvine Beowulf Cluster.

5 Results

5.1 Software Implementation Experience

5.1.1 Installation

Installation of the CFDACE+ package was done under both Windows and Linux platforms. The Windows version of CFDACE+ was installed onto local lab PCs for CFD problems that required minimal computing power. For example, CFDACE+ has been used regularly to simulate cold flow aerodynamics simulation in various geometries using RANS. Most of these cases have about 600,000 unstructured cells and can be finished in a couple of hours. Geometries are imported from Solidworks or AutoCAD and grids are generated with CFDGeom. The simulations are run with CFDACE and results viewed with CFDView. If the result is not satisfactory, Solidworks is used to modify the design for next simulation. Using this approach, hundreds of simulations for various applications have been run. Until now, the Windows version of CFDACE+ works very well on local PCs with Windows XP Professional. The installation of the Windows version is also straightforward, and no major problems were encountered.

After learning about the MPC cluster, a Linux version of CFDACE+ with the parallel run capability was obtained and installation proceeded on the MPC Beowulf Cluster for LES simulation and other CFD cases demanding large amount of computation. This process was not straightforward and it took many weeks to get the software working. The problem was mostly caused by the license installation. It was found that, unfortunately, the license installation tool `slm_admin` couldn't install the license properly. So a licensing file (`lservrc`) was obtained and put directly into the licensing directory, which was successful.

Another issue encountered during the Linux installation is that license must be installed on the head node of the MPC cluster. At first, another machine (UCICL local PC) was identified to serve as a license server for convenience and control. However, we encountered license-accessing problems because all the computing nodes in the cluster must get licensing information from the head node.

In summary, getting the software installed and operable on the LINUX cluster proved to be challenging and required considerable assistance from CFDRC. In contrast, the Windows version was installed on a local PC without any issues.

5.1.2 Parallel Operation

Operation in parallel mode presented some issues related to the PBS structure. Initially, parallel operation in PBS system was attempted by using a host file to tell CFDACE+ what nodes should be used. This often caused conflict since we this essentially bypassed the queuing system. After consulting with CFDRC support, `cfpace_mpi` was operated in the PBS environment with the parameter `- pbs`, which proved to be successful.

5.1.3 Run CFDACE on X terminal

Because Linux OS is not supported in the UCICL local area network, in order to communicate with the MPC cluster, X terminal simulator Exceed7.0 was used on the Windows XP Lab PC. However, several problems were encountered when running

CFDACE+ remotely on Exceed7.0. The major problem is that the X simulator doesn't support OpenGL graphic extension well. CFEDGEOM and CFEDGEUI crashed frequently when opening a grid file. XCFEDGEUI and XCFEDGEUI for X-window were both evaluated. These packages worked but the running speed was very slow in Exceed. To date, an acceptable method of running CFDACE+ smoothly in Exceed has yet to be identified. As a result, the DTF file is generated with the Windows version of CFDACE+ and then uploaded (via FTP) to the remote machine in the cluster. This is not very convenient because changing the DTF file cannot be done in the Linux environment remotely. Hopefully, CFDACE+ will improve the compatibility with X window simulator and run better in Exceed.

5.1.4 Time Limit

As mentioned above, the UCI MPC cluster administration applies a 24 hour time limit to each task submitted. As a result, only CFD cases that require less than 24 hours can be implemented. We have worked with the MPC cluster administrator to add nodes to the cluster, which will then be dedicated to our CFD simulations without limit. However, despite placing the order for 4 Opteron nodes, a desire on the part of the system administrator to consolidate orders to further reduce pricing led to delays and to date the units have not been received. Unfortunately, this time limit administration issue limited our ability to solve reacting flow LES cases and impacted our abilities to get good time resolution on cold flow cases. Once the machine orders are released and the units installed, we can then begin study of these more difficult cases.

5.2 Measured Flowfield

The diagnosis region in the combustor is shown in Figure 10. The full size of captured image is 107.9mm(H)*109.0mm(V) and the diagnosis region size is 74.3mm(H) * 98.5mm(V). The bottom of this region is 5.3mm above the bluff body face.

Figure 10. PIV Diagnosis Region in the combustor.

Tecplot v. 8.0 was utilized to conveniently manipulate and average the DPIV vector fields. Figure 11 shows the time averaged flowfield in the combustor as a result of averaging 100 images taken at a rate of 1 Hz. The results reveal an axisymmetric flow with strong recirculation as expected. The length of the recirculation zone is approximately 1 pipe diameter. Regions of high downstream velocity are apparent along the walls where the flow enters through the annulus between the bluff body and the wall.

Figure 11. Time averaged Flowfield

In contrast to the symmetric and well-behaved time averaged behavior, the typical instantaneous behavior, as shown in Figure 12, features considerable irregularities and departures from the time averaged flow, which is again expected.

Figure 12. Instantaneous Flowfield

5.3 Predicted Flowfield

Due to challenges outlined above, significant LES results in terms of total physical time were not obtained for this project. Both RANS cases and LES cases were

run on the MPC cluster, but due to time limits and configuration issues, the LES operation was only recently accomplished, and with less than desired total solution time.

Figure 13 shows the flowfield predicted using steady state RANS. The trends are again as expected.

Figure 13. RANS Flow Field (Velocity Magnitude)

The time-averaged LES flowfield is illustrated in Figure 14. It is noted that this average is only based on 1 msec of time, so it not truly “time averaged.”

Figure 14. LES Average Flow Field (Velocity Magnitude)

5.4 Comparison Between Predicted and Measured Flowfield

5.4.1 Time Averaged V Velocity

The time averaged velocities from RANS and DPIV are shown in Figure 15. The general trends are similar for each case, yet the RANS solution shows a larger and longer recirculation zone.

Figure 15. Time Averaged V velocity

To provide more quantitative comparison, Figure 16 presents time-averaged profiles along the diameter of the flowfield at different axial distances of 10, 20, 30, and 40 mm downstream of the bluffbody face. At 10 mm, the experimental results appear somewhat asymmetric and have a lower magnitude recirculating velocity compared to the simulation. By 20 mm, however, the agreement between the experiment and simulation is good and this agreement remains satisfactory at the 30 and 40 mm axial distances as well.

a) 10 mm

b) 20 mm

c) 30 mm

d) 40 mm

Figure 16. Comparison of RANS and DPIV Axial Velocities.

5.4.2 Time Resolved Axial Velocity

Figure 17 illustrates the instantaneous V velocity field (i.e., axial velocity) from the DPIV experiment and LES simulation. The flowfield is not driven by any external means, therefore the transient behavior is simply due to natural forces. Because the flow isn't driven, it is difficult to "synchronize" the snapshots from either technique. As a result, the comparison shown is not as illustrative as it might otherwise be. The results do show that the natural variation in the flowfield is captured by both DPIV and the LES and that the time-averaged view shown above in Figure 15 is only one interpretation of the flowfield. Time variant behavior at each point in the flow can be examined, but has not been at this point due to the limited LES total time (i.e., only 4 ms is available). To provide enough LES results, a much longer run time is needed, which is limited by the current 24 hour restriction on the cluster use.

Figure 17. Comparison of Flowfield Snapshots from DPIV and LES.

6 Conclusions and Recommendations

6.1 Conclusions

- The implementation of the parallel operation of the software required some unanticipated issues to be overcome. In addition, some non-technical issues (e.g., time limits) created a non-ideal situation in which to fully evaluate the utility of the software.
- Once the various licensing and operational issues were overcome, the parallel operation of the code was achieved and provided some interesting, albeit limited results.
- The steady state result (RANS) shows some good correspondence with the PIV time average result.
- LES result shows the instantaneous fluctuation of the flowfield that RANS cannot provide us. However, we cannot do more detailed quantized comparison since we only have limited LES results caused by our current computing resource.

6.2 Recommendation

- A solution to overcome the time restriction issue was implemented during the project through a strategy to obtain nodes dedicated to use for parallel operation, but due to the desire to “pool” our node order with others on campus, the order has still not been placed at the time of this document.
- Once the nodes are in place, additional LES solution should be obtained and other quantities compared to the DPIV results such as turbulence levels and vorticity.
- While the total airflow was accurately known, the actual inlet conditions used in the simulation may have differed from the experiment. Additional characterization of the inlet conditions could improve comparisons.

7 References

L.A. Ikioka, R.D. Brum, and G.S. Samuelsen (1983).
LES Validation Study: Impinging Jets in Confined Crossflow, Dr Adrian Spencer (2002)
CFDACE Modules Manuals, CFDRC (2002)

Appendix

DTF Files (RANS and some LES) (on CD).