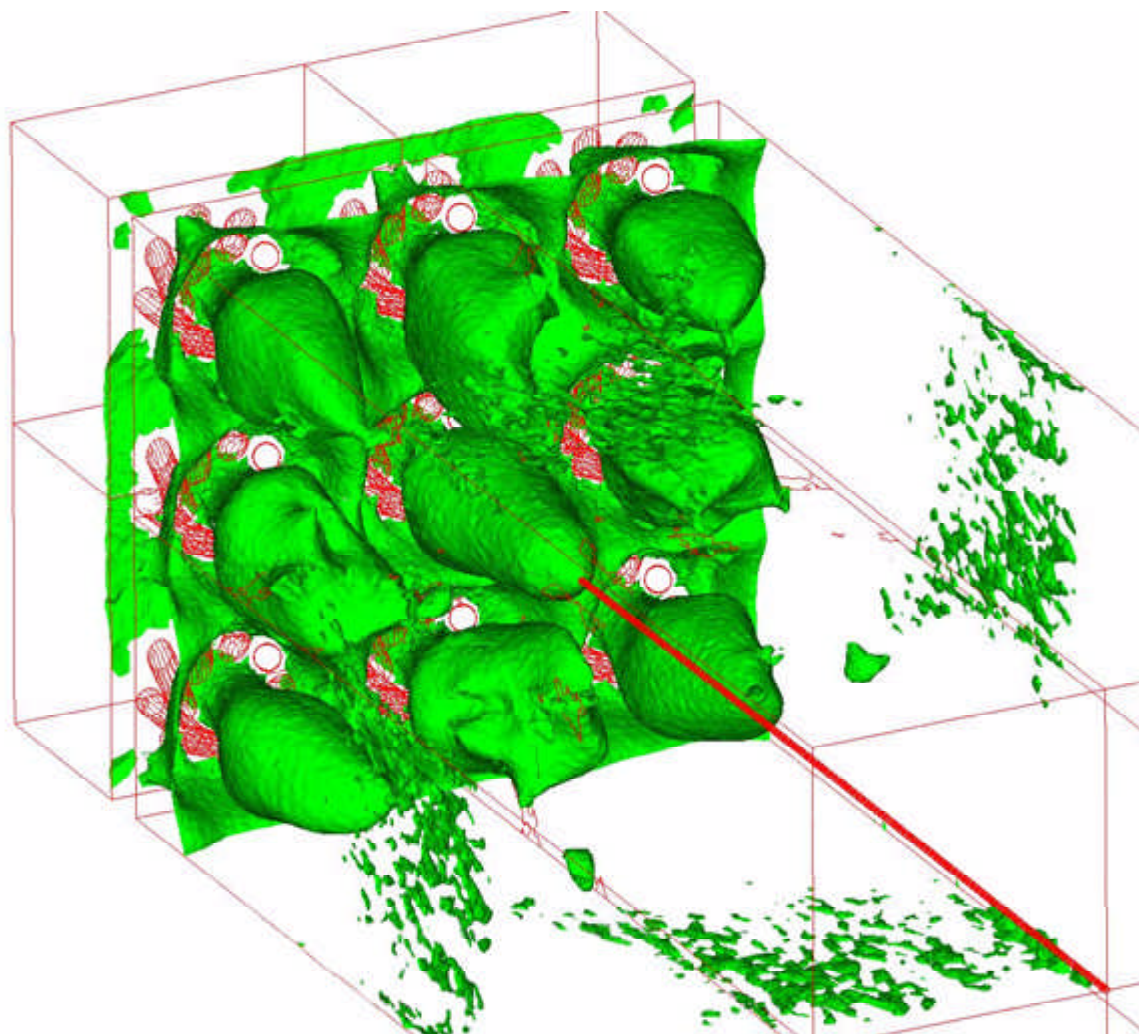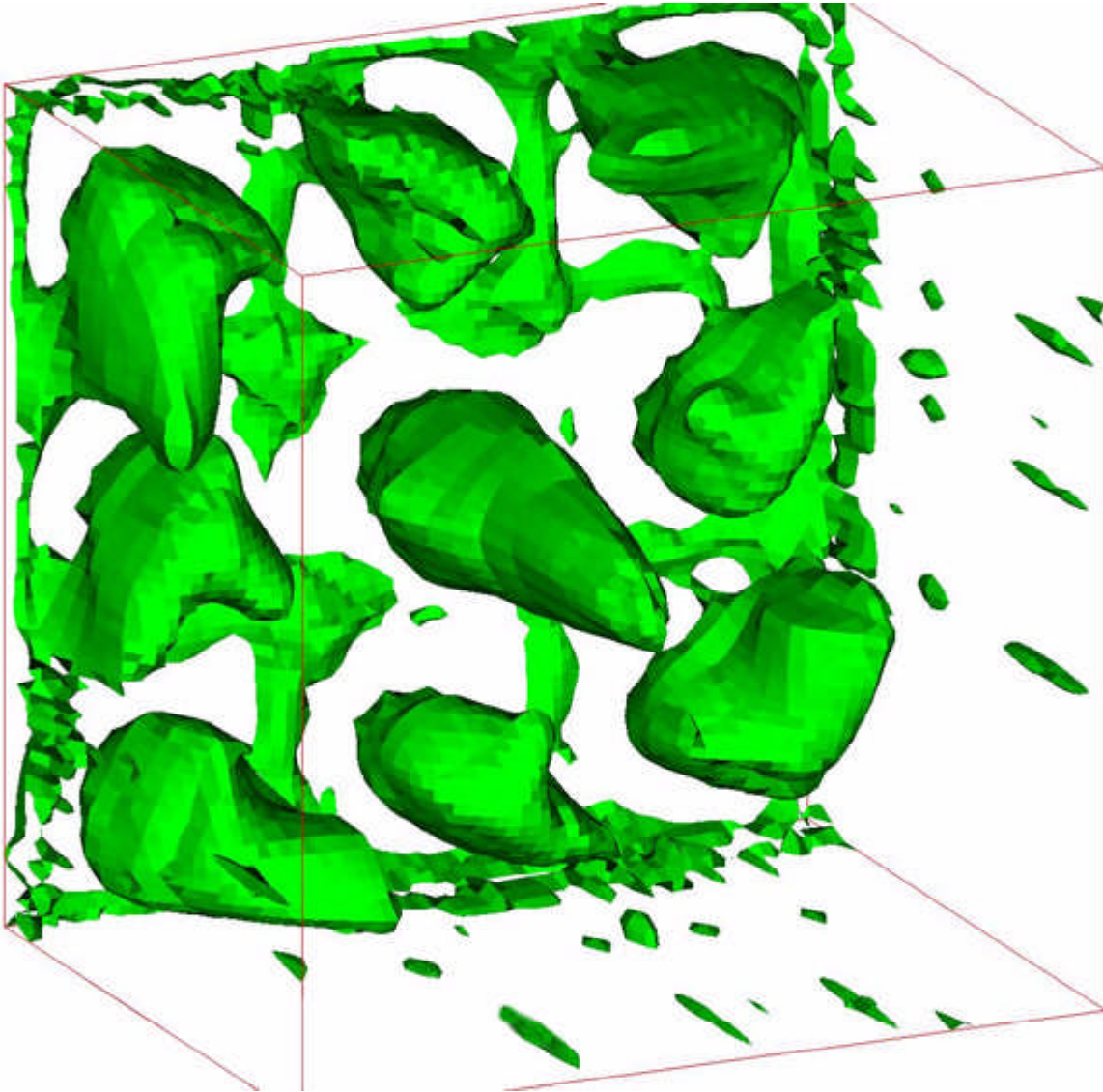# National Combustion Code Validated Against Lean Direct Injection Flow Field Data

*Top: NCC computations of the 35° co-rotating swirler. Pressure drop, 4 percent; atmospheric pressure; cubic, nonlinear k-*ε *turbulence model; low-Reynolds-number wall treatment. Bottom: Laser Doppler velocimetry measurements of the 35° LDI swirler. Static pressure drop, 4 percent; atmospheric pressure.*

Long description of figures The figures compare the recirculation zone by visualizing the isosurfaces calculated by the NCC (top) and measured by laser Doppler velocimetry (bottom). The two isosurfaces are nearly identical. This visualization shows that the NCC does compare well with experimental data.

Most combustion processes have, in some way or another, a recirculating flow field. This recirculation stabilizes the reaction zone, or flame, but an unnecessarily large recirculation zone can result in high nitrogen oxide (NO_x) values for combustion systems. The size of this recirculation zone is crucial to the performance of state-of-the-art, low-emissions hardware. If this is a large-scale combustion process, the flow field will probably be turbulent and, therefore, three-dimensional. This research dealt primarily with flow fields

resulting from lean direct injection (LDI) concepts, as described in Research & Technology 2001 (ref. 1). LDI is a concept that depends heavily on the design of the swirler. The LDI concept has the potential to reduce NO_x values from 50 to 70 percent of current values, with good flame stability characteristics. It is cost effective and (hopefully) beneficial to do most of the design work for an LDI swirler using computer-aided design (CAD) and computer-aided engineering (CAE) tools. Computational fluid dynamics (CFD) codes are CAE tools that can calculate three-dimensional flows in complex geometries. However, CFD codes are only beginning to correctly calculate the flow fields for complex devices, and the related combustion models usually remove a large portion of the flow physics.

The National Combustion Code (NCC) is a state-of-the-art CFD program designed specifically for combustion processes. The features of NCC pertaining to this study are (1) the use of unstructured grids (ref. 2), (2) massively parallel computing-with almost perfectly linear scalability (ref. 3), (3) a dynamic wall function with the effect of an adverse pressure gradient (ref. 4), (4) low-Reynolds-number wall treatment (ref. 5), and (5) a cubic nonlinear k-ε turbulence model (refs. 6 and 7). The combination of these features is usually not available in other CFD codes and gives the NCC an advantage when computing recirculating turbulent flows. These features need to be validated before the NCC is accepted as a design tool. The NCC was previously benchmarked for simple flows (ref. 8), and large-scale validations are being conducted (ref. 9).

The purpose of this study was to quantify how well the NCC calculates a turbulent, three-dimensional, recirculating flow field. The comparison was against three-dimensional laser Doppler velocimetry (LDV) measurements on a prototype LDI swirler (ref. 10). This configuration used nine groups of eight holes drilled at a 35° angle to induce swirl. These nine groups created swirl in the same direction, or a co-rotating pattern. The static pressure drop across the holes was fixed at approximately 4 percent. Computations were performed on one-quarter of the geometry, because the geometry is considered rotationally periodic every 90°. The final computational grid used was approximately 2.26 million tetrahedral cells, and a cubic nonlinear k-ε model was used to model turbulence. The NCC results were then compared with time-averaged LDV data. The LDV measurements were performed on the full geometry, but four-ninths of the geometry was measured. One-, two-, and three-dimensional representations of both flow fields are presented. The NCC computations compare both qualitatively and quantitatively well with the LDV data, but differences exist downstream. The comparison is encouraging and shows that NCC can be used for future injector design studies.

Find out more about the research of Glenn's Combustion Branch, http://www.grc.nasa.gov/WWW/combustion/

References

1. Tacina, Robert R., et al.: Sector Tests of a Low-NO$_x$, Lean, Direct-Injection, Multipoint Integrated Module Combustor Concept Conducted. Research & Technology 2001, NASA/TM--2002-211333, 2002, p. 116.

2. Stubbs, Robert M.; and Liu, Nan-Suey: Preview of the National Combustion Code. AIAA Paper 97-3114, 1997.

3. Quealy, A., et al.: National Combustion Code: Parallel Implementation and Performance. NASA/TM--2000-209801 (AIAA Paper 2000-0336), 2000.

4. Shih, Tsan-Hsing, et al.: Generalized Wall Function for Complex Turbulent Flows. NASA/TM--2000-209936, 2000.

5. Chien, Kuei-Yuan: Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model. AIAA J., vol. 20, no. 1, 1982, pp. 33-38.

6. Shih, Tsan-Hsing, et al.: Modeling of Turbulent Swirling Flows. NASA TM-113112, 1997.

7. Shih, Tsan-Hsing; Liu, Nan-Suey; and Chen, Kuo-Huey: A Non-Linear k-Epsilon Model for Turbulent Shear Flows. AIAA Paper 98-3983, 1998.

8. Chen, K.-H, et al.: Benchmark Test Cases for the National Combustion Code. AIAA 98-3855, 1998.

9. Iannetti, Anthony C.; and Chen, Kuo-Huey: Initial Comparison of National Combustion Code Simulations to Experimental Gas Turbine Combustor Data Using Various Chemistry Modules. AIAA-2000-0330, 2000.

10. Iannetti, Anthony C., et al.: Towards Accurate Prediction of Turbulent, Three-Dimensional, Recirculating Flows With the NCC. AIAA Paper 2001-0809, 2001.

Glenn contact: Anthony C. Iannetti, 216-433-5586, Anthony.C.Iannetti@nasa.gov
Author: Anthony C. Iannetti
Headquarters program office: OAT
Programs/Projects: Propulsion and Power, SEC, UEET, HPCCP