N94-23043

NUMERICAL INVESTIGATION OF TWO- AND THREE-DIMENSIONAL HEAT TRANSFER IN EXPANDER CYCLE ENGINES

Robert L. Burch and Fan-Bill Cheung
Propulsion Engineering Research Center
and
Department of Mechanical Engineering
The Pennsylvania State University
University Park, PA 16802

SUMMARY:

The concept of using tube canting for enhancing the hot-side convective heat transfer in a cross-stream tubular rocket combustion chamber is evaluated using a CFD technique in this study. The heat transfer at the combustor wall is determined from the flow field generated by a modified version of the PARC Navier-Stokes Code, using the actual dimensions, fluid properties, and design parameters of a split-expander demonstrator cycle engine. The effects of artificial dissipation on convergence and solution accuracy are investigated. Heat transfer results predicted by the code are presented. The use of CFD in heat transfer calculations is critically examined to demonstrate the care needed in the use of artificial dissipation for good convergence and accurate solutions.

TECHNICAL DISCUSSION:

The expander cycle has recently been considered one of the most desirable cycles for space propulsion applications. A liquid rocket engine operating on this cycle makes additional use of the fuel as the coolant. The performance of such an engine usually depends strongly on efficient combustor heat transfer to the coolant (i.e., fuel). By enhancing the rate of heat transfer in the combustion chamber, an increase in the chamber pressure can be achieved, leading to improved engine performance. One way to increase convective heat transfer is the use of round or oval coolant tubes, running longitudinally, to form the inner wall of the combustion chamber. This increases heat transfer by increasing the wall surface area, while providing relatively uniform heat fluxes. Berkopec [1992] reported wall heat transfer increases of up to 30% for some specific applications. One possible way to further increase the wall heat transfer is through tube canting in a coolant tube-lined combustion chamber. In this design concept, the coolant tubes are not parallel to the flow of combustion products. Rather, they are oriented at an angle α to the flow direction. With this arrangement, the surface contour varies continuously in the downstream locations. The boundary layer that would have been built up in the crevices is likely to be broken up or "tripped" by the angled tubes. This may result in increased local mixing of the flow, thus offering a potential for heat transfer enhancement over that of a non-canted tubular combustion chamber.

To evaluate the tube canting concept, a detailed numerical investigation of the flow and heat transfer in the combustion chamber of a split expander demonstrator cycle engine is conducted using a modified version of the PARC Navier-Stokes code. Both turbulent and laminar two- and three-dimensional flow fields were investigated. For the turbulent flow case, turbulence generated in the flow is approximated by using the Baldwin and Lomax algebraic model. The working fluid was assumed to be a non-reacting combustion gas using the physical properties that are consistent with the actual case.

The portion of the engine which is of interest includes the combustion chamber before the throat area. The inner wall of the combustion chamber is made up of 350 oval coolant tubes. The interior shape of the engine wall is three-dimensional but periodic in the azimuthal direction. Using the assumption of axisymmetry, the wall geometry used in the two-dimensional cases is a longitudinal wall cut made from the three-dimensional configuration. This should provide the greatest effect or influence on heat transfer (i.e., best case) for any wall geometry or helical tube angle, since previous studies had shown that helical ribs at 90 degrees had the greatest effect on heat transfer [Gee and Webb 1980]. The two-dimensional cases provide information concerning overall trends which should hold true in the three-dimensional cases, such as the effect of flow separation on heat transfer and the rib angle required for flow separation. The results of the two-dimensional study were reported at the previous symposium [Burch and Cheung 1992]. Additional two- dimensional calculations were performed to provide reference data for comparison with the three-dimensional results. In the three-dimensional cases a periodic boundary condition was used allowing calculation of one sector (1/350 of the total flow field) made up of one coolant tube interval rather than the whole flow field.

The three-dimensional grid generation program written for this study uses a generated two-dimensional grid for each required downstream or streamwise location. A Steger-Sorenson [1979] elliptical smoother refines the grid produced by an algebraic grid model. This helps to insure grid orthogonality at the wall as well as allowing control of grid clustering along the wall. A close-up of the wall surface contour is shown in Figure 1.

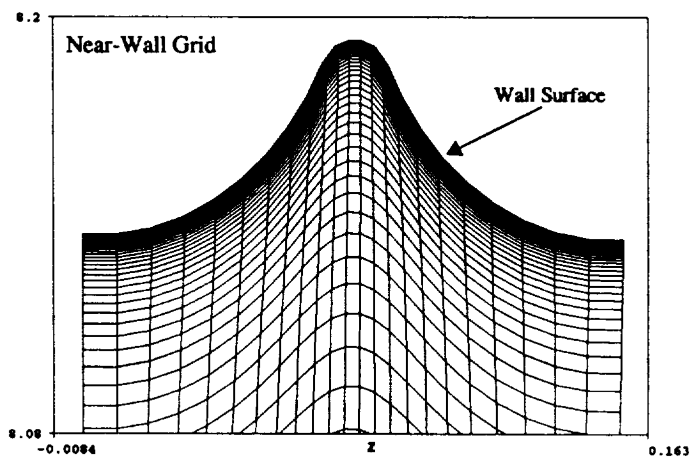

Figure 1. The near-wall region from an axial cut showing the wall grid structure

A three-dimensional test case was successfully run using a smooth wall combustion chamber to ensure the two- and three-dimensional versions of the PARC code produced the same results, once the implementation of the periodic boundary condition was corrected. The results of three-dimensional smooth wall case should be the same as the axisymmetric two-dimensional case. This case also allowed for check out of the three-dimensional versions of the programs used to calculate the wall heat transfer as well as other supporting post-processing programs.

The initial three-dimensional calculations used a zero degree canting angle, i.e., the coolant tubes run streamwise, in the baseline case. The turbulent heat transfer results ran counter intuitive, they do not seem to be physically correct. Because of resource limitations on the PSC CRAY YMP C-90, a 1/6 length test case was developed to run in-house using a IBM RS6000. Upon further testing it was reaffirmed that the results were not physically acceptable. The error in the solution was initially speculated to be a problem with the turbulence model, the artificial dissipation, or a combination of both. The same grid was subsequently run with laminar flow in order to remove the effects of the turbulence model from the solution. Again, similar heat transfer results were obtained. The greatest heat flux was occurring in the crevice or gap between the two adjacent coolant tubes. Upon further examination, there seemed to be a correlation between the grid spacing at the wall and the heat flux; the larger the distance of the first point off the wall, the lower the heat transfer. The correlation seemed to override the seemingly natural heat flux suppression due to the larger thermal boundary layer developing in the crevice. This correlation was confirmed using two-dimensional test cases. The calculated wall heat flux was found to be dependent on the wall grid spacing rather than the amount of grid stretching.

The variation of the wall heat flux with wall grid spacing in the laminar case was traced to solution sensitivity to the artificial dissipation. The PARC codes use central differencing scheme which requires artificial dissipation to provide high frequency damping and good convergence properties [Turkel and Vatsa 1990]. Central difference schemes experience odd and even point decoupling that must be damped to achieve satisfactory convergence [Swanson and Turkel 1987]. The PARC codes use a Jameson-style scalar dissipation model with an anisotropic directional scaling factor as suggested by Swanson and Turkel [1987]. To demonstrate the solution sensitivity due to the artificial dissipation, several cases were run with varying values of the fourth-order dissipation coefficient. On a global scale the thermal boundary layer appears to be independent to the amount of fourth-order dissipation used except for a temperature overshoot that accompanies case A. This aspect can be seen in Figure 2a. However, close to the wall, the tem-

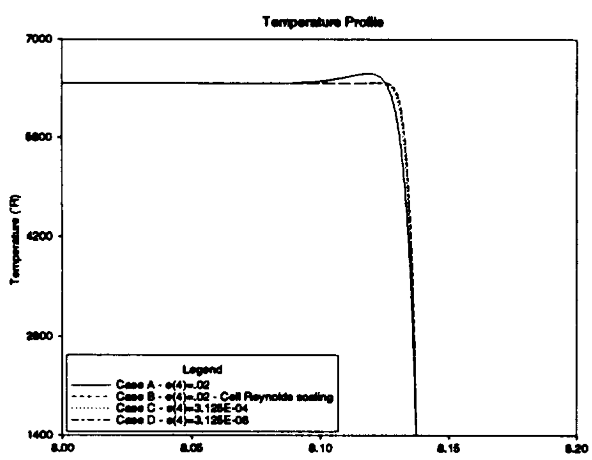

Figure 2a. The Thermal boundary layer

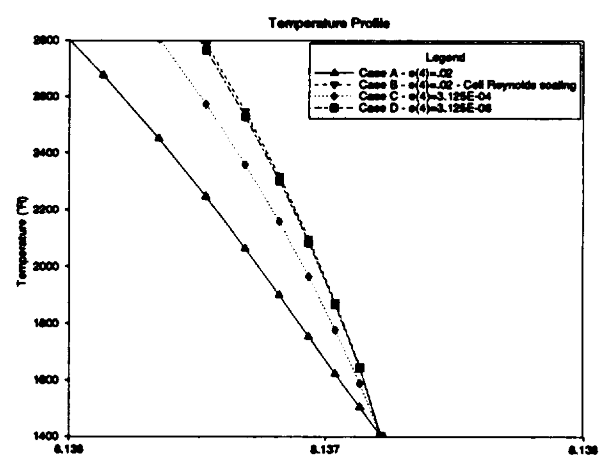

Figure 2b. The near-wall region of the thermal boundary layer.

perature profile shows marked differences in the wall temperature gradient, dT/dX, as seen in Figure 2b. Case A uses the recommended values for the fourth-order artificial dissipation coefficient ($\varepsilon^{(4)}$=.02). Cases C and D use significantly smaller dissipation coefficient (3.125E-04 and 3.125E-06, respectively). Case B uses a program option which scales the artificial dissipation coefficient with the cell Reynolds number. This effectively shuts the artificial dissipation off in the radial direction since there is very little flow in this direction, but at the price of a much slower rate of convergence. This option was used for all the previously reported two-dimensional results [Burch and Cheung 1992]. For three-dimensional calculations, this scaling method did not provide enough artificial dissipation for efficient convergence and physically acceptable results.

This sensitivity has been documented by Jameson [1993], Swanson and Turkel [1992,1993], and Turkel and Vatsa [1993]. Alternatives to the previously described implementation of the artificial dissipation have included the scaling of the artificial dissipation in the direction normal to the surface boundary with the local Mach number [Swanson and Turkel 1987] and the implementation of the matrix artificial dissipation as suggested by Swanson and Turkel [1992]. The use of the local Mach number scaling did reduce the sensitivity of the solution to the artificial dissipation but did not eliminate it. The matrix artificial dissipation model is under evaluation using the two-dimensional test cases.

Turbulent cases are less sensitive to the artificial dissipation swamping the physical dissipation due to the fact that a larger physical dissipation exists due to the turbulent viscosity. Therefore, any improvement in the artificial dissipation as seen in the laminar test cases should be acceptable for turbulent test cases. With the sensitivity of the artificial dissipation resolved, attention can be given to the turbulence model if perceived problems persist in the solution. The adequacy of the Baldwin-Lomax turbulence model when used with this type wall surface contour can be evaluated. Particular attention must be given in the proper implementation of the local turbulent length scales.

REFERENCE

Berkopec, F. D., 1992, "Space Vehicle Propulsion," NASA-Lewis Research Center, March 17 - Presentation.

Burch, R. L. and Cheung, F.-B., 1992, "Numerical Study of Heat Transfer in Expander Cycle Engines," Fourth Annual PERC Symposium, NASA - Marshall Space Flight Center, Huntsville, AL, pp. 85-88.

Gee, D. L. and Webb, R. L., 1980, "Forced Convection Heat Transfer in Helically Rib-Roughened Tubes," *Int. J. Heat Mass Transfer*, Vol. 23, pp. 1127-1136.

Jameson, A., 1993, "Artificial Diffusion, Upwind Biasing, Limiters and their Effect of Accuracy and Multigrid Convergence in Transonic and Hypersonic Flow," *AIAA Paper 93-3359*.

Steger, J. L. and Sorenson, R. L., 1979, "Automatic Mesh-Point Clustering Near a Boundary in Grid Generation with Elliptic Partial Differential Equations," *Journal of Computational Physics*, Vol. 33, pp. 405-410.

Swanson, R.C. and Turkel, E., 1987, "Artificial Dissipation and Central-Difference Schemes For the Euler and Navier-Stokes Equations," *AIAA Paper 87-1107*.

Swanson, R.C. and Turkel, E., 1992, "On Central-Difference and Upwind Schemes," *Journal of Computational Physics*, Vol. 101, pp. 292-306.

Swanson, R.C. and Turkel, E., 1993, "Aspects of a High-Resolution Scheme for Navier-Stokes Equations," *AIAA Paper 93-3372-CP*.

Turkel, E. and Vatsa, V. N., 1990, "Effect of Artificial Viscosity on Three Dimensional Flow Solutions," *AIAA Paper 90-1444*.