

Hypersonic Turbulent Reacting Flow Simulation of Fire II Re-entry Vehicle

Krishnendu Sinha* and Siva Krishna Reddy†

*Department of Aerospace Engineering
Indian Institute of Technology Bombay, Mumbai 400 076, India*

Abstract

Prediction of hypersonic turbulent flow field with non-equilibrium thermo-chemistry around re-entry vehicles is a challenging task. The predictions error are high due to the uncertainties in the turbulence models. In this paper, flow field around the Fire II re-entry module is computed using the Reynold-averaged Navier Stokes method. Five species air chemistry and a two-temperature vibrational relaxation model is used to compute the reacting flow around the vehicle. The freestream conditions correspond to the lowest altitude, and therefore highest Reynolds number, for which in-flight measurements are available. One-equation Spalart-Allmaras and two-equation $k-\omega$ models are used for turbulence closure, and the model predictions are compared with in-flight measurements of afterbody pressure and heat transfer rate. In addition, the effect of turbulence and thermo-chemistry on the flow field are studied by comparing controlled simulations at identical free stream conditions.

Introduction

Atmospheric re-entry vehicles are subjected to large aero-thermal loads. During the later part of their descent the flow around the vehicle, especially in the wake region, may transition to turbulence. This can significantly increase the heat transfer to the vehicle walls. Predicting turbulent heating rate is challenging due to the uncertainties of the turbulence models, and the inaccuracies are generally compensated by a large safety factor in the design of heat shields¹.

Brown² computed the flow around the MP1 re-entry configuration of Hollis et al. Performance of several turbulence models was investigated against heat transfer data on the model afterbody and sting. The flow was found to transitions at the flow reattachment region on the sting. However, it is not clear how the sting interference affects the wake and consequently the heat transfer on the afterbody. The current paper will study the transitional/turbulent flow around the Fire II re-entry vehicle, where the in-flight heating data do not have any sting interference effect. Also, the Reynolds number at the conditions considered here is about twice that in Ref. 2. Transition to turbulence is therefore expected to occur earlier on the vehicle, possibly even on the forebody.

Sinha³ use Reynolds-averaged Navier-Stokes method along with the $k-\omega$ and Spalart-Allmaras turbulence models to simulate the Fire II re-entry flowfield. The focus was to understand the behavior of turbulence models at the low Reynolds number and high Mach number conditions typical of re-entry conditions. It was found that the SA turbulence model needs to be modified in order to avoid unrealistic heating prediction on the afterbody. In addition, shock-unsteadiness modification to the $k-\omega$ model was required to avoid over-amplification of TKE at the bow shock. There were differences in the predictions of the two models, specifically in the size of separation bubble and afterbody pressure. Although the temperature in the Fire II flow are high enough for chemical reactions to occur, the simulations in Ref. 3 assume perfect gas and the effect of thermo-chemistry is neglected. As a result, the aerodynamic predictions could not be validated against flight data.

The objective of the current paper is to compute reacting turbulent hypersonic flow around the Fire II re-entry module and compare the aerodynamic predictions against in-flight measurements. A second, and equally important, objective is to gain an in-depth understanding of the physio-chemical phenomena involved in the flow. The flow physics is a combination of gas dynamics, turbulence/transition and non-equilibrium thermo-chemistry. The effect of each phenomena is isolated and analyzed by comparing controlled numerical studies.

* Assistant Professor, Department of Aerospace Engineering, Indian Institute of Technology Bombay. Member AIAA.

† Graduate student, Department of Aerospace Engineering, Indian Institute of Technology Bombay.

For example, the differences between perfect gas and reacting computations highlight the effect of chemistry on the flow prediction. The resulting aerodynamic characteristics of the vehicle in each case are also presented.

The paper is organized as follows. A brief description of the Fire II vehicle geometry and the associated physical phenomena are presented below. This is followed by the simulation methodology along with the grid generation technique. Results include description of RANS flow field solution and surface properties, as well as comparison of laminar vs. turbulent and reacting vs. perfect gas computations to study the effect of different phenomena. Effect of different turbulence models is also presented. The aerodynamic predictions are finally compared to flight data.

2. Fire II Configuration

Project Fire flights⁴ were conducted to investigate the heating environment on a blunt-nosed, Apollo-shaped vehicle entering the earth's atmosphere at a velocity in excess of the escape velocity. This configuration is preferred over wind-tunnel experiments due to the absence of sting effects. Sinha *et al.*⁵ simulate the conditions (see Table I) at a point in the later part of the trajectory which corresponds to one of the lowest altitudes and highest Reynolds numbers for which measurements are available.

Figure 1, taken from Ref. 4, shows a schematic of the Fire II vehicle. The spherical forebody consisted of a multi-layer configuration made of three protective phenolic asbestos heat shields sandwiched between three beryllium calorimeters. The first two calorimeters and their associated heat shields were jettisoned during the flight at predetermined deceleration loads. The third heat shield was not ejected and this gives the shape of the vehicle through the later part of the re-entry that is of interest in this work. The afterbody is conical with a 66° included angle. The C-band antenna at the base is replaced by a flat base for simplicity, which gives the geometry of the vehicle used in the current work (Fig. 1).

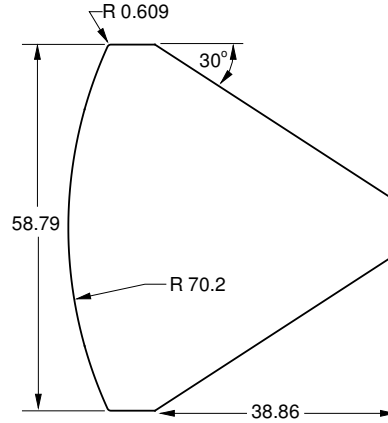


Figure 1. Fire II re-entry vehicle geometry used for the current simulations. All dimensions are in cm.

3. Simulation Methodology

The chemically reacting turbulent hypersonic flow around the Fire II re-entry configuration will be simulated by solving the Reynolds-averaged Navier-Stokes equations along with the species conservation equations and the thermal non-equilibrium model. A five-species air chemistry model (N_2 , O_2 , NO , N and O) will be used with three dissociation reactions and two exchange reactions.⁶ The rate constants are evaluated using curve fits to experimental data by Park.⁷ Thermal non-equilibrium is modeled using a rotational-translational temperature and a single vibrational temperature. An additional equation is solved for the vibrational energy of the mixture with the Landau-Teller vibrational relaxation source terms given in Ref. 8. The $k-\omega$ model of Wilcox⁹, and the Spalart-Allmaras model¹⁰ are used for turbulence closure.

The equations are discretized using the finite volume approach. Inviscid fluxes are computed using a modified (low-dissipation) form of the Steger-Warming flux splitting approach, and the turbulence model equations are fully coupled to the mean flow equations. The method is second order accurate both in stream-wise and wall normal directions. The viscous fluxes and the turbulent source terms are evaluated using second order accurate

central differencing and the implicit Data Parallel Line Relaxation is used to obtain steady-state solutions. The details of the formulation are given in Ref. 11.

The grid topology used in the simulations is shown in Fig. 2. The outer boundary follows the shape of the bow shock and extends up to 2.5 D downstream of the vehicle base. The flow at this point is found to be supersonic and therefore the location of the exit boundary is not expected to alter the flow solution in the vicinity of the vehicle. A typical grid consists of about 160 points in the wall parallel (i) direction and 120 points in the wall normal (j) direction respectively. Grids with larger number of points are also used in some cases. The grid in the i -direction is designed such that the points are clustered in the nose stagnation region, and the expansion corners at the shoulder and base. A fine mesh is needed at the nose stagnation point to get a smooth variation of the heat transfer rate. The grid in the j -direction is refined both in the vicinity of the shock wave as well as in the near-wall region. Grid lines are also carefully aligned to the shock wave to get a grid converged solution.

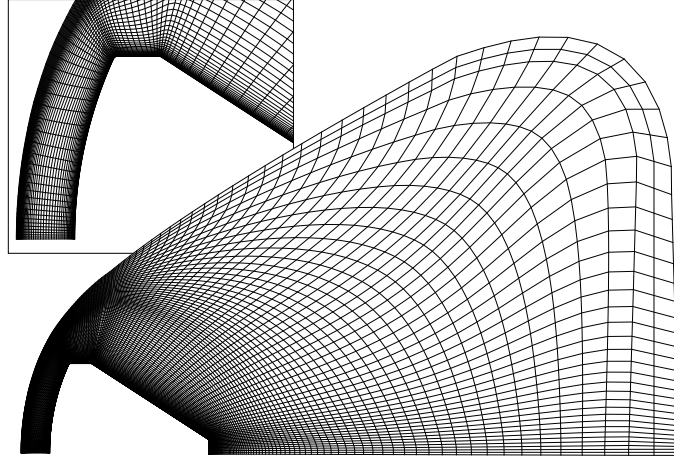


Figure 2. Computational domain and grid used in the simulations. The inset shows a magnified view of the grid near the vehicle, where every second point is shown in each direction.

Results

Figure 3a shows a RANS solution computed using the SA turbulence model in terms of the Mach number contours (top half). The prominent features like bow shock and flow expansion at the corners can be easily identified. The temperature in the stagnation region is about 5700 K and it varies less than 5% on the forebody. The flow on the entire forebody is subsonic with the Mach number approaching unity close to the first expansion corner. Eddy viscosity is negligible compared to molecular viscosity in the nose stagnation region, and it grows to about $\mu_t/\mu \sim 5$ near the shoulder. The strong flow expansion at the shoulder results in thinning and relaminarization of the boundary.

A magnified view of the shoulder region in Fig. 3b shows that the boundary layer remains attached through the two successive expansion corners. Flow separation on the conical frustum is found to generate a mild shock. The resulting separation bubble extends up to about 0.4 diameter downstream of the base. The shear layer enclosing the recirculation region coalesce at the neck and a recompression shock wave is generated. The neck is marked by high temperature and pressure, and is followed by flow expansion in to the far wake. The recirculation bubble is characterized by a single toroidal vortex and the Mach number is subsonic in this region. The lower half of Fig. 3a shows the eddy viscosity level normalized by the local molecular viscosity. $\mu_t > 20\mu$ in the recirculation region and its value increases through the neck up to about 300μ due to mean flow compression.

The mixture temperatures and composition are plotted along the stagnation streamline in Fig. 4 to study the shock and boundary layer regions. Temperature peaks at about 8500 K immediately downstream of the shock, followed by a rapid decrease due to transfer of translational energy into dissociation of N_2 and O_2 molecules. The vibrational temperature shows a gradual rise to about 6200 K followed by decrease to match the translational-rotational temperature. The flow attains equilibrium downstream of the shock where the temperature is about 5700 K. Oxygen is mostly dissociated in this region with mass fraction of O at 0.22. Nitrogen dissociation is relatively small with post-shock equilibrium mass fractions of N_2 and N at 0.71 and 0.05 respectively. NO is

formed behind the shock to a maximum level of 8%, but it dissociates in the equilibrium region. In the vicinity of the wall, both translation-rotation and vibration temperature decrease to the cold wall temperature specified at 553.3 K. The resulting recombination leads to drop in C_N and C_O , and increase in the concentration of O_2 and N_2 . C_{NO} is also found to reach about 0.05 in this region. The variation of fluid properties across the boundary layer along the forebody show a trend similar to that discussed above.

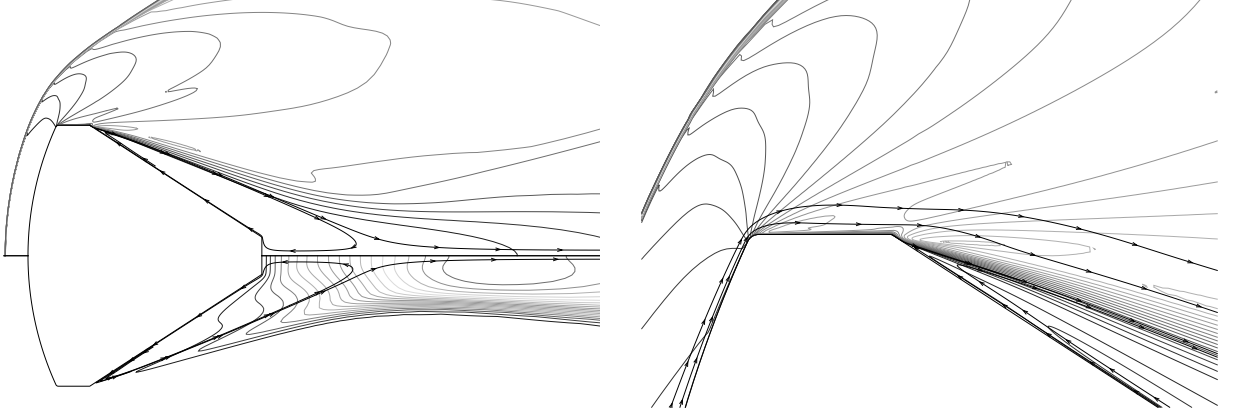


Figure 3. Turbulent flow simulation around the Fire II re-entry vehicle: (a) Mach number (top half) and normalized eddy viscosity (bottom half) distributions, and (b) normalized pressure contours in the shoulder region. Representative streamlines are shown to identify the recirculation bubble and separation location.

Contrary to the equilibrium flow on the forebody, the thermo-chemistry is frozen as the flow expands around the shoulder. Fig. 5 plots the variation of the flow composition and temperature along a wall normal line. Note that the y -coordinate is plotted on the vertical axis, which extends from just outside the boundary layer to freestream. The trend seen in the boundary layer is similar to that observed near the nose stagnation point in Fig. 4b. Behind the bow shock, the variation in the properties is similar to that observed on the stagnation streamline. Temperature shows a local peak followed by dissociation and vibrational excitation. However, there is large discrepancy between T and T_v values. Temperature decreases significantly due to flow expansion at the corner. Vibrational temperature does not follow this trend, which leads to a large discrepancy between T and T_v . This non-equilibrium effect is emphasized by plotting the variables along a streamline (Fig. 5b). Vibrational temperature and species concentrations show a mild variation at the onset of flow expansion followed by constant values. This is due to the low flow density in this region that reduces the chemical and thermal relaxation times significantly, resulting in frozen flow.

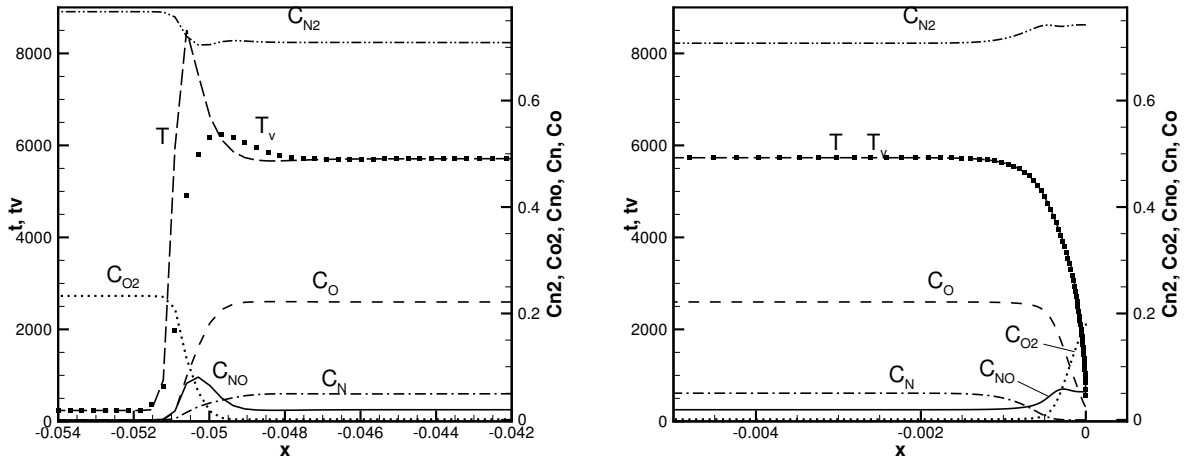


Figure 4. Variation of temperatures and species mass fraction along the stagnation streamline: (a) at the shock, and (b) in the near-wall region.

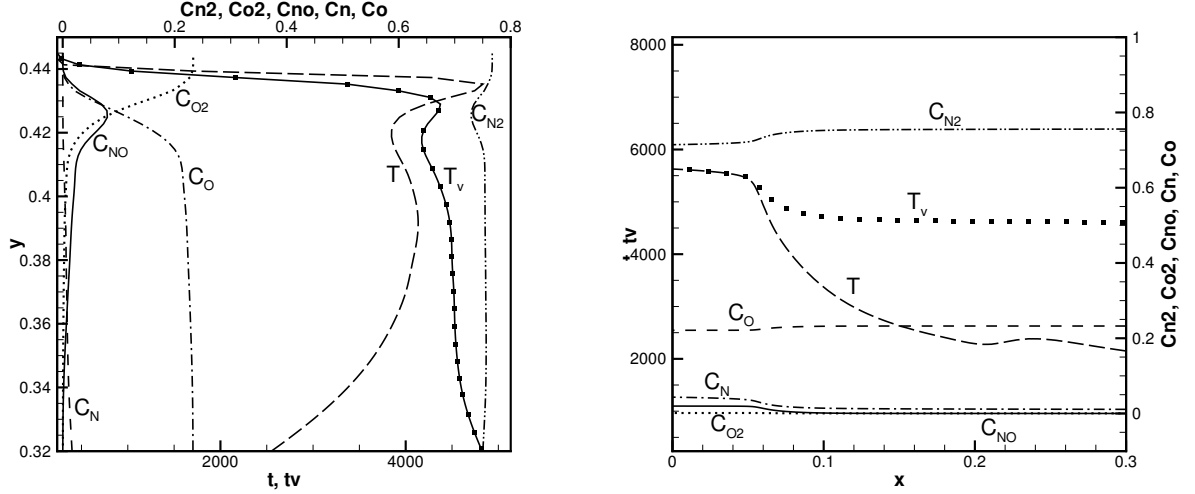


Figure 5. Variation of temperatures and species mass fraction along (a) wall-normal line at the shoulder, and (b) a streamline.

The composition of the gas in the inviscid flow outside the wake is primarily determined by the low pressure and temperature (around 2000 K) prevailing in this region. Here, the gas mixture is primarily composed of N_2 and O . The flow in the recirculation bubble is comparatively hotter with maximum temperature around 6000 K. This results in partial dissociation of N_2 . The mass fraction of N_2 is between 0.76 and 0.71, and that of N is less than 0.05. O_2 and NO constitute less than 6% of the gas mixture each and C_O varies between 0.16 and 0.23.

Figure 6 presents the surface properties obtained using the RANS equations. The forebody wall pressure follows a parabolic profile with maximum at nose stagnation point and a rapid drop due to expansion on the shoulder. Heat transfer rate is mostly uniform on the forebody with a maximum value of 220 W/cm² at the stagnation point. Results from a laminar solution are also included for comparison. The turbulent heating rate deviates from the laminar value on the later part of the forebody. Difference between the two solutions is less than 4% because of the low level of eddy viscosity in this region. At the expansion corners, the heating rate increases due to thinning of the boundary layer followed by a sharp decrease as the temperature drops through the expansion fan. Turbulence level also decreases at flow expansion resulting in identical heating rates in the laminar and turbulent solutions immediately downstream of the first corner. Further transition on the shoulder results in higher heating rate (by up to 83% just before the second expansion corner) than the laminar case.

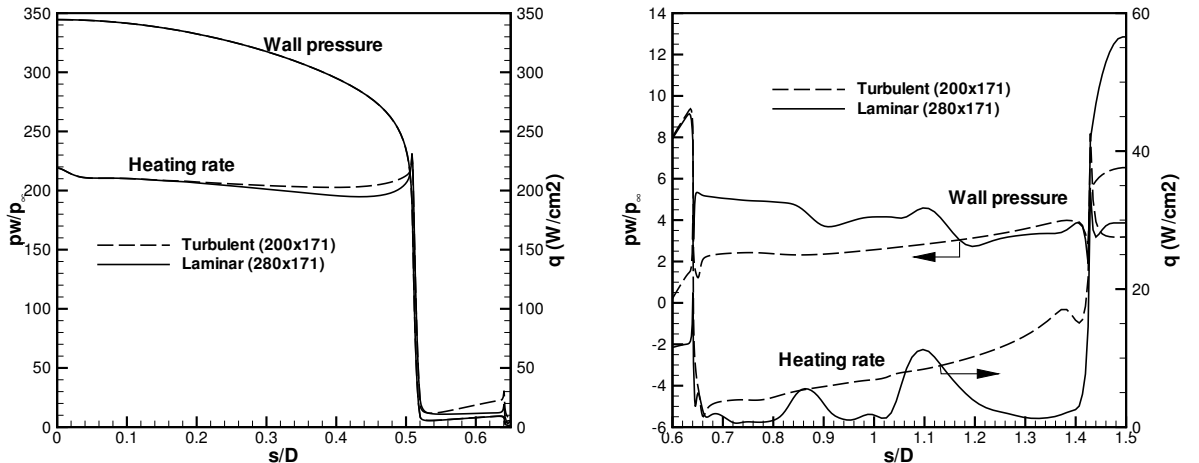


Figure 6. Comparison of laminar and turbulent results of normalized pressure and heat transfer rate on the (a) forebody and (b) afterbody of the Fire II vehicle at Mach 16.

The pressure and heating rate in the afterbody separation bubble are presented in Fig. ‘fig:ptura’b. Flow expansion on to the conical afterbody results in a sharp decrease in pressure at $s/D \simeq 0.65$. A similar pressure drop due to the expansion of the reversed flow at the base corner is at $s/D \simeq 1.42$. The heat transfer rate is found to rise and then fall rapidly at these expansion corners. In between the two expansion corners, the turbulent pressure and heating rate show a monotonic increase from the separation point at the shoulder to the base corner. The base stagnation point pressure and heating rate are less than 1% and 15% respectively of the nose stagnation values. Afterbody laminar results presented in the figure are discussed below.

The above results were computed using 100 wall-parallel points on the forebody and shoulder with clustering at the nose stagnation region and the expansion corners at the shoulder. In the wall-normal direction, fine grid is used in the boundary layer and in the vicinity of the bow shock. The grid is significantly coarser in between them. The outer boundary is very close to the bow shock and the grid lines are carefully aligned with the shock wave. This is essential to avoid large numerical error at the stagnation point which may result in unphysical heating rate in this region. The first grid point next to the wall is at 10^{-6} m which results in $y_2^+ \leq 0.1$. Wall parallel grid refinement is focused at the expansion corners, where additional clustering is not found to alter the solution (including surface heating rate) significantly.

The sensitivity of the afterbody results to grid point density and mesh quality is presented next. From a baseline grid of 160x150 points, the grid in the i and j -directions are refined independently to arrive at a grid converged solution. Refining the entire grid in both directions simultaneously leads to significant increase in computational time. In addition, zonal refinements are carried out to identify the critical regions, where high grid density is required. In the j -direction, wall normal spacing of 10^{-6} m, which is equivalent to $y_2^+ \leq 0.94$ was found to be sufficient for getting the correct separation location and wall heat transfer rate. The size of the wake and the associated pressure and heating rate on the conical frustum is sensitive to accurate resolution of the free shear layer. Successive refinement (shown in Fig. 7a) led to the minimum number of grid points (171) required in the j -direction. In the wall parallel direction, a fine grid near the separation location along with a low stretching factor on the conical frustum is necessary for grid convergence. The results are however not very sensitive to the grid spacing at the rear stagnation point, the expansion corner at the base, and the neck region. Careful refining near the separation location leads to a grid converged solution on a 280x171 mesh. Figure 7b shows that the difference in wall pressure between the two finer meshes is less than 4% on the afterbody. Heating rates vary less than 2% in this case.

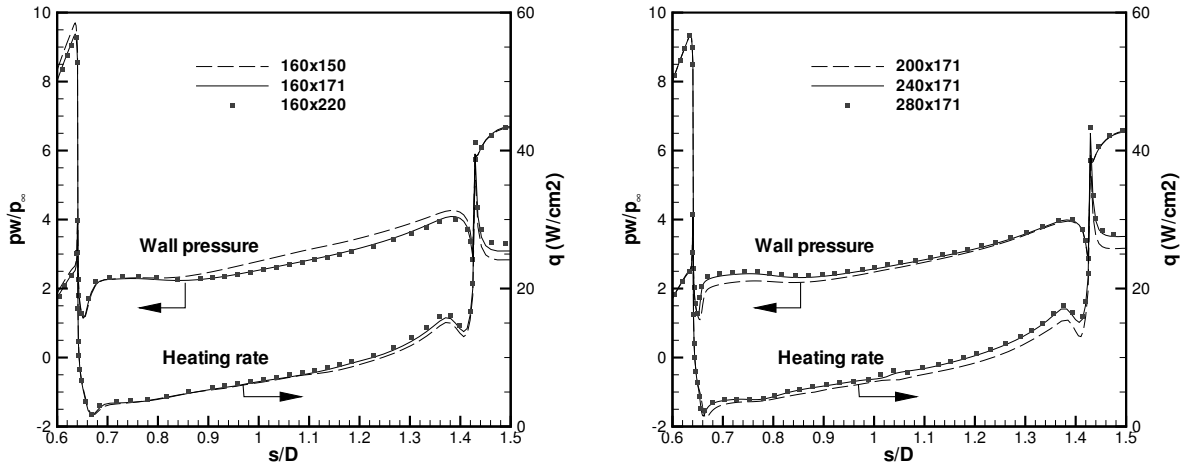


Figure 7. Normalized pressure and heat transfer rate on the afterbody computed using successively refined grids: (a) wall-normal refinement and (b) wall-parallel refinement.

Effect of turbulence

To study the effect of turbulence on the flowfield, the above RANS results are compared to laminar solution computed at identical conditions. The forebody flow field as well as the expansion on to the shoulder and

afterbody is almost identical to the turbulent solution presented in Fig. 3. The main difference is in the afterbody flowfield. The laminar solution has a much larger recirculation region which extends to about 1D downstream of the base. Also, in contrast to the turbulent solution, the reversed flow along the axis becomes supersonic. A normal shock is therefore formed at the base to slow down the flow before it reaches the rear stagnation point. The laminar recirculation region is marked by multiple vortices originating from the primary, secondary and tertiary separation along the conical frustum. These vortices are found to oscillate, coalesce and separate over time resulting in an unsteady flow in this region. The size of the recirculation region as well as the location of the base shock is also found to change with time. The flow outside the wake is not affected much. It is not possible to get a steady state solution. Note that the high eddy viscosity level ($\mu_t/\mu > 20$) in the turbulent solution dissipates the smaller vortices and results in a single toroidal vortex in the separation bubble.

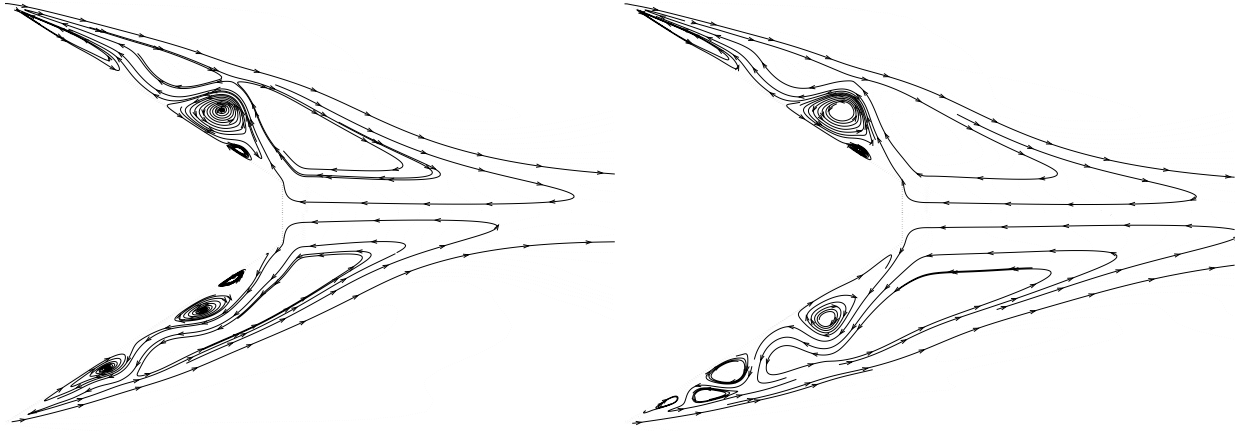


Figure 8. Afterbody flow field shown in terms of temperature contours and representative stream lines. The top half of each figure represents the solution obtained on the baseline 160x171 grid and the bottom solutions are obtained on (a) 160x220 and (b) 240x171 meshes.

Figure 8 presents laminar results obtained on three computational grids. In addition to the baseline grid of 160x171 points, data from two other grids 160x220 and 240x171, obtained by refinement in the j and i directions respectively, are presented. The flow pattern in the recirculation region is sensitive to the grid point density. The number of vortices and the size of the separation bubble decrease with grid refinement in the j -direction. Opposite trend is observed with i -direction grid refinement. Grid converged solution could not be achieved, which points to the fact that the flow may be three-dimensional and unsteady. Three-dimensional perfect gas simulation for the current configuration corroborates this fact.

Afterbody surface properties obtained from the laminar simulation are shown in Fig. 6b. In contrast to the turbulent results, the laminar pressure and heat transfer rate on the conical frustum have multiple peaks and minima. The low values of pressure correspond to the vortex cores and the high pressure regions are local flow separation and reattachment points. By comparison low heat transfer rate is at flow separation and high local heating is found at reattachment regions. As the grid is refined, the number of undulations and their magnitude is found to increase. This trend is similar to the vortex pattern described above.

At the beginning of the separation bubble, pressure in the laminar solution is higher than the turbulent value. This is because of a smaller bubble in the turbulent case leading to larger flow expansion at the corner. Laminar pressure decreases along the frustum whereas the turbulent pressure increases downstream such that their values are comparable towards the end of the conical afterbody. At the base, presence of a normal shock in the laminar solution results in pressure level twice that in the turbulent case. The laminar solution also has a stronger expansion of the reversed flow at the base corner than the turbulent case.

The turbulent heat transfer rate increases along the afterbody from values comparable to the laminar case near the separation point to about 850% higher at the end of the frustum. The higher heat transfer rate in the turbulent case is mainly because of higher temperature in the core of the recirculation bubble, which determines the edge temperature to the near-wall temperature gradient. The eddy viscosity is comparable to the molecular viscosity ($\mu_T/\mu \leq 5$) in this near-wall region and it enhances the heat transfer to the wall further. At the base,

the effect of the base shock in the laminar solution is compensated by higher temperature in the turbulent case to yield comparable heating rates in the two simulations. An interesting point to note is that at the separation point near shoulder, turbulent heating shows the same trend as skin friction coefficient, that is to decrease monotonically. By comparison, laminar heating rate increases at separation location. A pressure rise is also observed in both solutions due to the separation shock.

The maximum temperature in the laminar recirculation bubble is about 3000 K, compared to 6000 K in the turbulent solution. As a result, there is negligible dissociation of N_2 , and the mass fraction of NO is relatively low. Mass fraction of O_2 varies between 0.06 and 0.10, whereas that of O is between 0.12 and 0.15 in the recirculation region. There is more dissociation of O_2 at the neck and base stagnation region because of the relatively higher temperature (around 4000 K) prevailing in these regions.

Comparison of turbulence models

Wilcox's $k-\omega$ model is used to compute the Fire II flowfield and the solution is compared with the SA results. The $k-\omega$ results are obtained on a 240×171 computational grid with wall normal spacing of 2.5×10^{-7} m, which is equivalent to $y_2^+ = 0.25$ or lower. Comparison of temperature contours in Fig. 9a show that the forebody flow pattern is identical in the two solutions. Afterbody temperatures are also similar, with somewhat higher temperature in the $k-\omega$ wake. Fig. 9b compares eddy viscosity levels in the two cases. Note that free-stream molecular viscosity is used as a common reference for normalization. Eddy viscosity is higher in SA solution which leads to a smaller recirculation bubble. As a result, pressure on the conical afterbody is lower in the SA

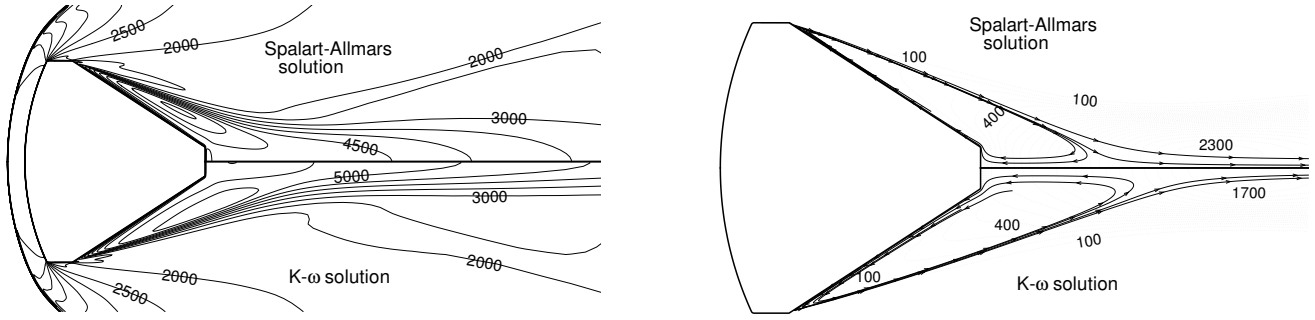


Figure 9. Comparison of SA and $k-\omega$ solution computed for the Fire II flow field: (a) temperature contours, and (b) Normalized eddy viscosity. Representative streamlines are shown to demarcate the recirculation bubble.

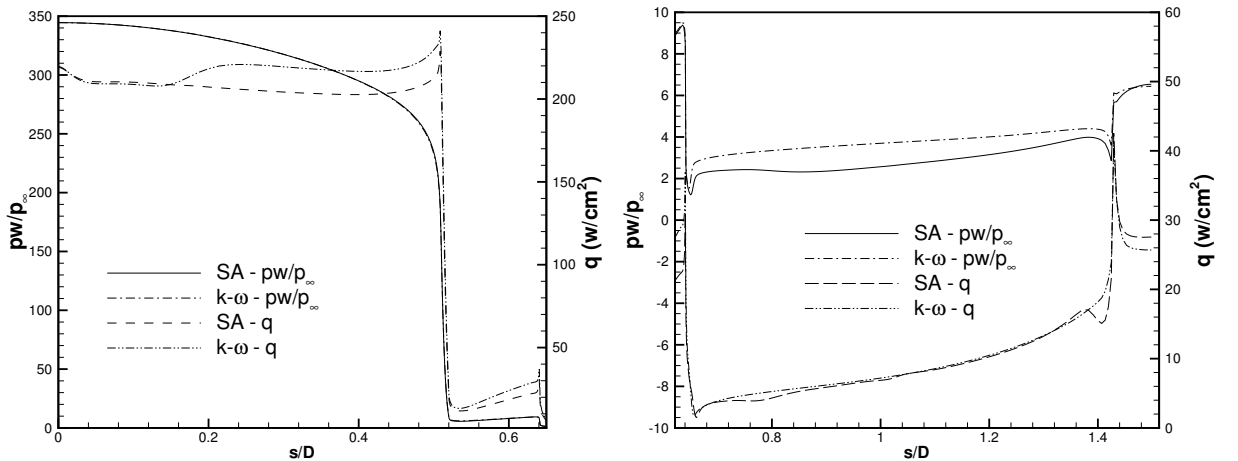


Figure 10. Fire II surface properties obtained using the SA and $k-\omega$ turbulence models: (a) forebody, and (b) afterbody results.

solution than in $k-\omega$ (see Fig. 10b). Shear layer angle, measured with respect to the conical frustum, is also higher in $k-\omega$ by about 2 degrees. Afterbody heat transfer rates in the two cases are identical (see Fig. 10b). This is because both temperature distribution in the separation bubble and the eddy viscosity levels in the near wall region are similar. On the forebody, only difference between the two solutions is in the heating rate (see Fig. 10a). $k-\omega$ shows a distinct jump at about $s/D = 0.2$ that corresponds to boundary layer transition and thus results in higher heating rate compared to SA.

Effect of chemistry

The effect of finite rate thermo-chemistry on the flow pattern and the resulting aerodynamic predictions is discussed by comparing chemically reacting and perfect gas simulations using the $k-\omega$ model. The trends observed in the corresponding SA solutions are similar to $k-\omega$ and are not presented for the sake of brevity. Perfect gas results are taken from Ref. 3, where a careful grid refinement study is presented. Fig. 11a shows the temperature and normalized turbulent viscosity distribution in the two flowfields. Reacting gas solution is characterized by lower temperature and higher density in majority of the flow field compared to perfect gas case. Higher density ratio across the bow shock results in smaller shock stand-off distance for reacting case by a factor of 0.59. On the afterbody, the local Re is higher for reacting case, which delays separation on the conical frustum and predicts a smaller recirculation zone in the reacting simulation. In the perfect gas solution flow separates at the expansion corner, where as for reacting case separation occurs downstream of shoulder, as can be seen from

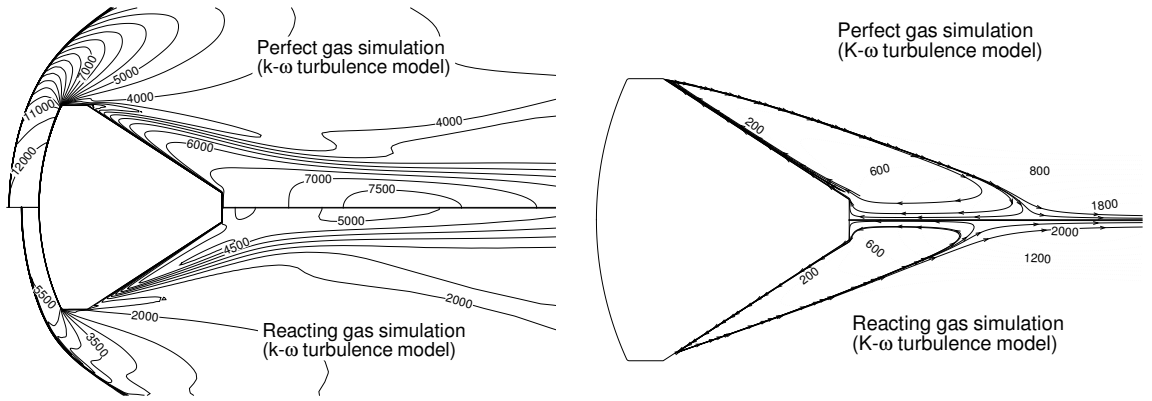


Figure 11. Comparison of perfect gas and reacting flow solutions computed using the $k-\omega$ turbulence model: (a) temperature contours, and (b) eddy viscosity distribution normalized by freestream molecular viscosity. Representative streamlines are shown to demarcate the recirculation bubble.

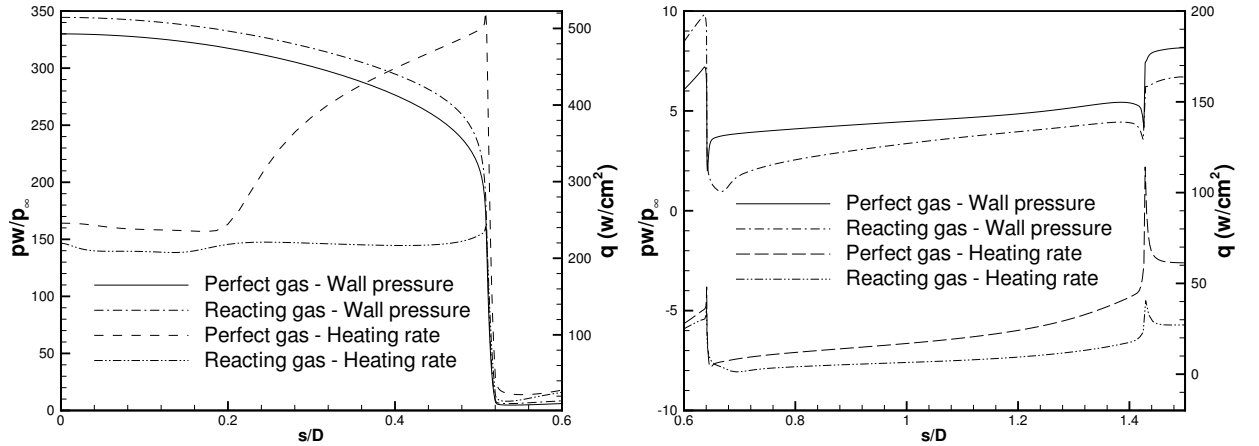


Figure 12. Fire II surface properties obtained in perfect gas and reacting simulations using the $k-\omega$ turbulence model: (a) forebody, and (b) afterbody results.

the streamlines in Fig. 11b. As expected, forebody pressure is higher and heating rate is lower in the reacting simulation as compared to perfect gas case. The rise in heat transfer rate at transition is much higher in the perfect gas solution. Similarly, pressure and heating rate on the afterbody are higher in the perfect gas simulation than the corresponding reacting values. This correlates well with the size of the separation bubble and temperature of the gas as discussed earlier.

Comparison with flight data

The Fire II vehicle afterbody was instrumented with a pressure sensor (at $s/D = 1.27$ and $\phi = 265^\circ$) and several gold calorimeters for surface temperature measurements. These calorimeters were placed at different axial locations along three circumferential rays ($\phi = 0, 120$ and 240 degs). Here, ϕ denotes the azimuthal angle around the vehicle. As noted by Slocumb,⁴ there is little variation in the temperature data with circumferential location. The system accuracy, including that of sensors and telemetry system, was approximately $\pm 20\%$ for the pressure data and about $\pm 28^\circ$ K for the temperature measurements.

Experimental measurements in supersonic base flows yield uniform base pressure in the separation bubble.¹² Similar trend is expected to be valid at higher Mach numbers, and therefore the pressure data at $s/D = 1.27$ can be taken to be representative of the entire conical frustum. This is indicated by the dotted line in Fig. 13a. The in-flight measurement is lower than all the different computations, with reacting gas simulation using SA turbulence model being the closest over the entire conical frustum. Perfect gas and laminar predictions are found to be significantly higher than the data.

Comparing the heating rate data in Fig. 13b, we see that the laminar results are far lower than flight data except for the local peaks in heat transfer. Even an average value will considerably underpredict the measurements. By comparison, RANS results are closer to the data, with values lower than measurement at the beginning of the recirculation bubble. The RANS prediction monotonically increases to exceed the data in the later part of the conical frustum. Also, there is little variation between the two turbulence models tested in this paper. Perfect gas results are higher than the reacting gas computation by about a factor of two, and they grossly overpredict the heating rate measurements.

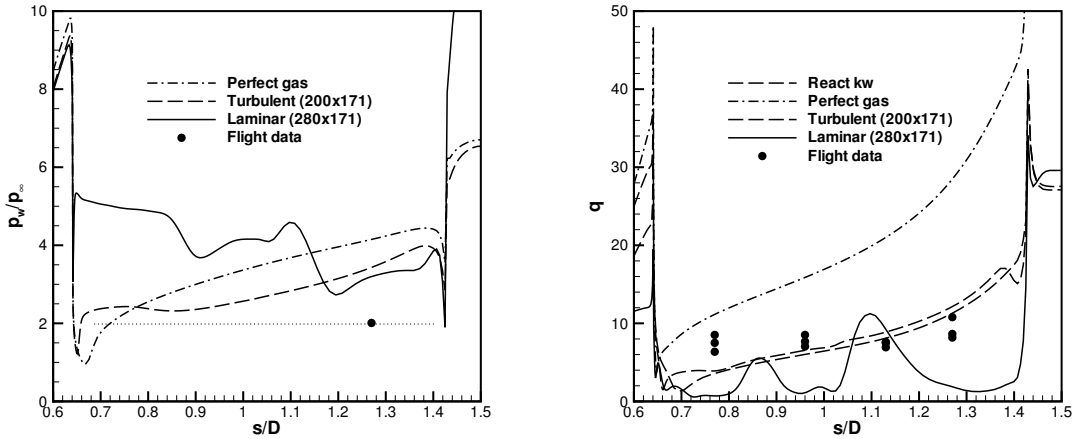


Figure 13. Comparison of computed surface pressure and heat transfer rate with in-flight measurements.⁴

Conclusions

Flow field around the Fire II re-entry module is computed using Reynold-averaged Navier Stokes method. Five species air chemistry and a two-temperature vibrational relaxation model is used to compute the reacting flow around the vehicle. The freestream conditions correspond to the lowest altitude, and therefore highest Reynolds number, for which in-flight measurements are available. One-equation Spalart-Allmaras and two-equation $k-\omega$ models are used for turbulence closure. $k-\omega$ model yields a distinct jump in heat transfer rate at the forebody transition location. On the afterbody, both model predictions of heating rates are similar and they compare well

with in-flight data. By comparison, laminar computations yield unsteady heat transfer peaks due to multiple vortices in the separation bubble. The effect of non-equilibrium thermo-chemistry on the flow pattern and aerodynamic predictions is studied by comparing perfect gas results at identical freestream conditions. Perfect gas simulation grossly overpredict the in-flight pressure and heating rate measurements.

Acknowledgments

The authors would like to thanks Santhosh Kumar, currently a graduate student at Stanford University, for his contributions towards grid generation for the current simulations.

References

- [1] Wright, M.J., Loomis, M., and Papadopoulos, P. . “Aerothermal Analysis of the Project Fire II Afterbody Flow”. *Journal of thermophysics and heat transfer*, 17(2), 2003.
- [2] Brown, J.L. “Turbulence model validation for Hypersonic flows”. *AIAA paper*, 2002-3308, 2002.
- [3] Sinha, K. “RANS simulation of Fire II re-entry configuration”. *International Symposium on Shock Waves*, 2005.
- [4] Slucomb, T.H. “Project Fire II afterbody Temperatures and Pressures at 11.35 Kilometers per Second”. *NASA TM X-1319*, 1966.
- [5] Sinha, K., Barnhardt, M., and Candler, G.V. “Detached Eddy Simulation of Hypersonic Base Flows with Application to Fire II Experiments”. *AIAA Paper*, 2004-2633, 2004.
- [6] M.J. Wright. “*A Family of Data-Parallel Relaxation Methods for the Navier-Stokes Equations*”. PhD thesis, Dept. of Aerospace Engineering and Mechanics, University of Minnesota, June 1997.
- [7] C. Park. “two-temperature interpretation of dissociation rate data for n_2 and o_2 ”. *AIAA Paper*, 88-0458, Jan 1988.
- [8] W.G. Vincenti and C.H. Kruger. “*Introduction to Physical Gas Dynamics*”. Krieger Pub., Malabar, Florida, 1986.
- [9] Wilcox, D.C. “Reassessment of the Scale-Determining Equation for Advanced Turbulence Models”. *AIAA Journal*, 26(11), 1988.
- [10] Spalart and Allmaras. A one-equation turbulence model for aerodynamic flows. *AIAA Paper*, 92-0439, 1992.
- [11] Sinha K. and Candler G.V. Convergence improvement of two-equation turbulence model calculations. *AIAA Paper*, 98-2649, 1998.
- [12] Bourdon, C.J., and Smith, K.M., and Dutton, J.C., and Mathur, T. Planar visualization of large scale turbulent structures in axisymmetric supersonic base flows. *AIAA Paper*, 98-0624, 1998.