

Aerothermal Uncertainty Quantification of Deployable Entry Technologies Using Multi-Fidelity Modeling

Mario Santos* and Serhat Hosder†

Missouri University of Science and Technology, Rolla, MO 65409

Thomas K. West‡

NASA Langley Research Center, Hampton, VA, 23681

The objective of this work was to investigate the use of a co-Kriging based multi-fidelity modeling approach with a Monte Carlo uncertainty quantification analyses of surface heating on hypersonic inflatable aerodynamic decelerator vehicles and adaptable, deployable entry placement technology vehicles in Mars entry. A previously developed co-Kriging based multi-fidelity modeling approach was used to model the laminar and turbulent convective and radiative heat fluxes along the vehicle. Monte Carlo analyses of surface heat load for nine different vehicle nose radii was performed for both vehicles, assuming the same thermal protection system is used for both vehicles. The maximum heat loads and heat load uncertainties were found to occur at either the stagnation point or the vehicle shoulder. The maximum 95% confidence intervals for the total heat load for the hypersonic inflatable aerodynamic decelerator vehicle were found to be [-9.27, 7.81] % of the nominal value and [-6.09, 13.1] % of the nominal value for laminar and turbulent flow, respectively. The maximum 95% confidence intervals for the total heat load for the adaptable, deployable entry placement technology vehicle were found to be [-8.85, 7.22] % of the nominal value and [-9.33, 9.35] % of the nominal value for laminar and turbulent flow, respectively. The difference in heating uncertainties based on vehicle geometry between the two vehicle technologies was found to be minimal.

Nomenclature

a	Hicks-Henne amplitude coefficient	x	Data point
b	Hicks-Henne width coefficient	y	Model response
c	Hicks-Henne control point location	α	Angle of attack (deg)
H	Enthalpy (J/kg)	β	Vehicle azimuthal angle (deg)
h	Heat transfer film coefficient (kg/m ² s)	β_{sc}	Toroid tangency angle (deg)
k_{sc}	Scallop depth (cm)	γ	Specific heat ratio
ℓ	Running length (m)	$\hat{\gamma}$	Flight path angle (deg)
M	Mach number	δ	Half cone angle (deg)
$\dot{q}$	Heat flux (W/m ²)	ϵ	Emissivity
R_b	Base radius (m)	θ	Shoulder turn angle (deg)
R_n	Nose radius (m)	$\hat{\theta}$	Longitude (deg)
R_s	Shoulder radius (m)	ρ	Density (kg/m ³)
R_{sc}	Scallop radius (m)	σ	Stefan-Boltzmann constant (W/m ² K ⁴)
R_{tr}	Toroid radius (m)	ϕ	Surface inclination angle (deg)
Re	Reynolds number	$\hat{\phi}$	Latitude (deg)
r	Altitude (m)	Ψ	Correlation matrix
s	Offset running length (m)	$\hat{\psi}$	Heading (deg)
T	Temperature (K)		
V	Velocity (m/s)		

*Graduate Student, Department of Mechanical and Aerospace Engineering, Student Member AIAA.

†Professor of Aerospace Engineering, Department of Mechanical and Aerospace Engineering, AIAA Associate Fellow.

‡Aerospace Engineer, Vehicle Analysis Branch, Systems Analysis and Concepts Directorate, Member AIAA.

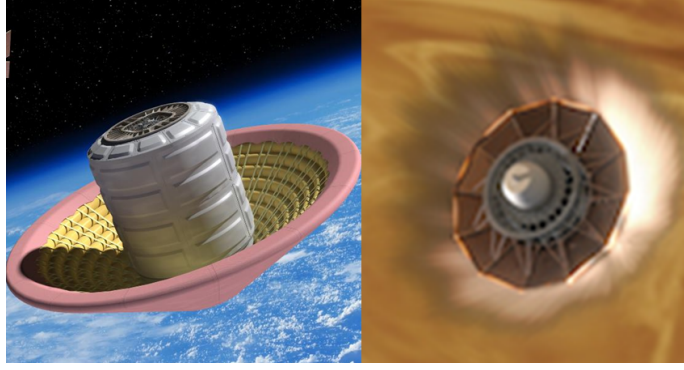


Fig. 1 HIAD [5] (left) and ADEPT [6] (right) conceptual designs.

I. Introduction

HUMAN exploration of Mars presents unique challenges to scientists and engineers in terms of mission and vehicle design. For human exploration, large payloads and human crew will need to be delivered to the planet surface at a large range of landing sites. Unfortunately, legacy rigid body vehicle designs are ill-fitted to meet mission requirements for these human exploration missions. To address these challenges, NASA has recently been developing deployable atmospheric decelerator technologies that will allow for an increase in mission design possibilities and flexibility over that provided by rigid aero-shells. These deployable entry technologies will allow for an increase in the number of feasible landing sites and an increase in the maximum deliverable payload mass to the surface of multiple planetary bodies in our solar system such as Mars, Venus, and Titan. These technologies include NASA's Hypersonic Inflatable Aerodynamic Decelerator (HIAD) [1, 2] and the Adaptable Deployable Entry and Placement Technology (ADEPT) [1, 3, 4] shown in Fig. 1 [5, 6]. Due to the large aerothermal loads placed on the vehicles, the entry, descent, and landing (EDL) phase is one of the most dangerous and critical phases of any interplanetary mission. Therefore, accurate predictions of the aerothermal loads experienced by the vehicles are essential for mission design and for the development of thermal protection systems (TPS) with minimal mass that can survive the EDL mission phase. The TPS also needs to be robust and reliable, which is achieved through extensive laboratory testing and through uncertainty quantification (UQ) integrated to the analysis and design.

Unlike most legacy entry vehicle designs, deployable entry vehicles have design specific attributes that can greatly increase the surface heat flux. HIAD vehicles have a flexible TPS that can deform under aerodynamic loads, causing surface scalloping that has been shown to increase convective heat flux [7]. ADEPT vehicles contain sudden surface changes at the rib locations, which can potentially lead to larger heat fluxes than axisymmetric vehicles with the same nose radius and cone angle. Thus, accurate prediction of these heat fluxes is paramount to the successful development of deployable entry technologies.

Due to the velocities, temperatures, and atmospheric compositions, entry conditions are often difficult or impossible to recreate in a laboratory setting. Thus, researchers often turn to computational modeling to predict the aerothermal loads on hypersonic entry vehicles in lieu of laboratory tests. However, due to the complex physics seen at these flow regimes (including turbulent flow, non-equilibrium thermo-chemistry, and radiation heat transfer), high-fidelity computational fluid dynamics (CFD) and thermal response simulations can be computationally very expensive. Therefore, CFD may not be practical for direct use in the analysis, design, and reliability assessment under uncertainty with traditional UQ approaches. This is because these approaches, such as Monte-Carlo analysis, generally require a large number of simulations making them unfeasible for use with CFD simulations. Multi-fidelity modeling is uniquely suited to solve this problem by creating low cost, high accuracy multi-fidelity surrogate models using a very limited number of high-fidelity CFD simulations. By replacing high cost, high-fidelity CFD models with low cost multi-fidelity models in a Monte Carlo UQ approach, UQ analyses that were computationally prohibitive due to high-fidelity CFD model evaluation costs can be performed quickly and with minimal computational resources.

To date, investigation of the use of multi-fidelity surrogate modeling for use in the study of entry vehicles has been very limited. Single fidelity surrogate modeling of high-fidelity CFD simulations has been previously used to reduce computational costs of robust aerothermal analyses of HIAD vehicles by Brune et al.[8–10]. Multi-fidelity modeling offers an alternative method that has the ability to achieve the accuracy of high-fidelity simulations but at a further reduced computational cost over single fidelity methods. Early work with applying multi-fidelity modeling

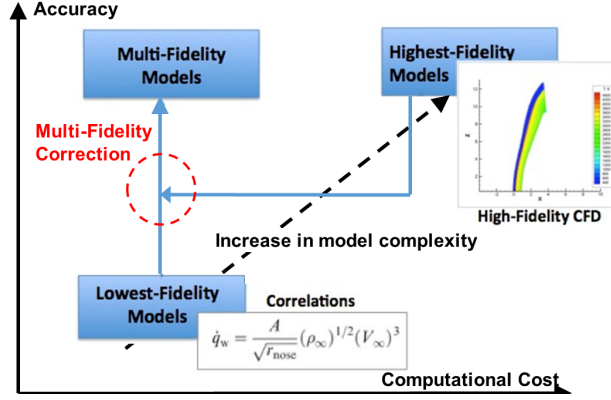


Fig. 2 Accuracy vs. computational cost.

to hypersonic vehicles was that of aerothermal anchoring utilized by the CBAERO software package [11]. Previous studies by the authors [12–14] demonstrated that a multi-fidelity approach can produce low cost, accurate predictions of laminar, turbulent, and radiative aerothermal loads on HIAD and ADEPT vehicles. These studies also demonstrated that the multi-fidelity approach was computationally less expensive than a single fidelity surrogate approach.

The main idea behind multi-fidelity surrogate modeling is to combine low-fidelity and high-fidelity models in a way that produces high accuracy surrogate models that are computationally efficient to evaluate. By using information from the low-fidelity models to inform us about the behaviour of the high-fidelity model, multi-fidelity modeling limits the number of high-fidelity model evaluations required to produce accurate surrogate models. A graphical representation of multi-fidelity modeling process is illustrated in Fig. 2. This figure shows that as model complexity increases, the associated computational cost also increases. By combining a low cost, less accurate engineering correlation with a more accurate high cost CFD model, multi-fidelity modeling can produce a surrogate that has a computational cost comparable to the engineering correlation but an accuracy comparable to the complex CFD model. These low cost multi-fidelity surrogate models can then be utilized in UQ analyses or design optimization studies that require large numbers of model evaluations. Multi-fidelity modeling has been used in many disciplines ranging from electrical to aerospace engineering where high-fidelity models are either computationally expensive or unknown. In the field of aerospace engineering, multi-fidelity modeling has been primarily used in low-speed applications for interpolation [11], wing and airfoil shape optimization [15–17], and uncertainty propagation [17–19]. There are currently several different multi-fidelity methods available that are appropriate for different applications. Adaptive and fusion techniques, such as space-mapping [15] and co-Kriging [16], are primarily used for interpolation and design optimization, while techniques using polynomial chaos expansion [18] and multi-fidelity Monte Carlo estimation [19] are used primarily for uncertainty quantification and propagation. This study will use the co-Kriging based multi-fidelity modeling approach previously developed for modeling convective and radiative heat flux on HIAD and ADEPT vehicles [12–14].

The objective of this work is to demonstrate and verify the use of co-Kriging based multi-fidelity (CKMF) surrogate models in UQ analyses of surface heat loads on HIAD and ADEPT vehicles during Mars entry. Vehicles with nose radii ranging from four to twenty meters will be analyzed and the surface heat fluxes modeled by the CKMF approach. The results from the UQ analyses will be used to make a relative comparison of surface heating based on vehicle geometry between the HIAD and the ADEPT. The multi-fidelity surrogate models for heat flux are constructed as functions of velocity, density, and nose radii to predict aerothermal loads. This research will utilize several computational tools of various fidelity levels such as the Sutton-Graves [20] with Krasnov [21] correlations (low-fidelity), West-Brandis radiative heating correlations (low-fidelity), and NASA’s LAURA CFD software using a 10-species non-equilibrium gas (high-fidelity) that will provide the necessary data to build the CKMF models.

The following section describes the HIAD and ADEPT geometries used in this study. Section III describes the Mars entry trajectory model. Section IV describes the low-fidelity, high-fidelity, and multi-fidelity heating models. Section V presents the results for each vehicle and the final section includes key conclusions from this work.

II. Vehicle Geometries

The first step of this study is to define and determine the geometry parameters of the HIAD and ADEPT vehicles. The HIAD vehicle consists of a solid spherical nose cap forward of the payload section, surrounded by concentric

inflatable toroids. A flexible TPS is stretched over the forward face of the toroids and is attached to the solid nose cap. The toroids remain deflated, stowing the TPS, until the vehicle begins the EDL phase, at which point the toroids are inflated, deploying the TPS for atmospheric entry. An illustration of a HIAD can be seen in Fig. 3. Similarly, the ADEPT vehicle concept also consists of a solid spherical nose cap forward of the payload section which is also surrounded by a flexible TPS. However, instead of inflatable toroids, the ADEPT can mechanically deploy or stow the flexible TPS using several ribs attached to the solid nose cap. An illustration of the ADEPT can be seen in Fig. 4.

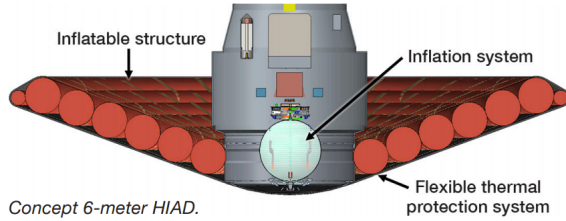


Fig. 3 HIAD vehicle diagram[22].



Fig. 4 ADEPT vehicle diagram[23].

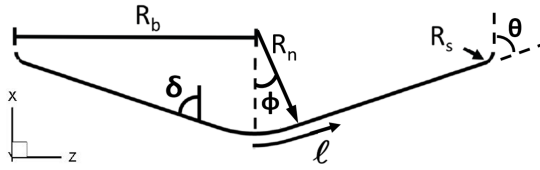


Fig. 5 Shared vehicle geometry parameters.

guide for this study. This scalloping parameterization technique is illustrated in Fig. 6. In this technique, a toroid tangency angle, β_{sc} , is defined relative to the normal of the undeformed TPS. A scallop of radius R_{sc} is then inscribed that is tangent to the neighboring toroids at these locations. The maximum distance between the scallop and the undeformed TPS is defined as the scallop depth, k_{sc} . For this study, a toroid radius, R_{tr} , of 0.35 m is used along with a tangency angle of 10° . This results in a scallop depth of 3.06 cm being used in this study. In actual flight, the scallop depth may change depending on the flight conditions and surface pressure, however a maximum expected scallop depth is used in this study as a constant through the trajectory points to give a conservative prediction of the surface heat flux.

B. ADEPT Specific Vehicle Parameters

The most significant difference in vehicle shape of the ADEPT when compared to the HIAD is that the ADEPT is not axisymmetric due to the rib system used to deploy the TPS. This also causes increased heat flux at the rib locations due to the sharp changes in the surface orientation at the ribs. The ADEPT vehicles used in this study have twelve ribs to deploy the flexible TPS, with each rib having a width of 20 cm. The ribs are distributed at every thirty degrees about the center axis. These parameters were chosen using a baseline case from a structural study of large diameter ADEPT vehicles by Peacocke et al. [26], and an example ADEPT geometry used in this study is illustrated in Fig. 7. Notice that for this geometry, there is a plane of symmetry at every 15° in the azimuthal, β , direction starting at a rib center line. For this study, it is assumed that the transition from the solid spherical nose cap to the flexible TPS occurs at the

Although the difference in TPS deployment systems leads to key differences in the HIAD and the ADEPT vehicle shapes, both vehicles share the same geometric parameters that define the overall basic vehicle shape. These shared geometric parameters describing the vehicle shape are the nose radius, R_n , base radius, R_b , shoulder radius, R_s , half-cone angle, δ , and shoulder turn angle, θ . These vehicle shape parameters are shown in Fig. 5. The HIAD and ADEPT geometries used in this study are based on the large diameter Mars entry vehicles described in NASA's Mars entry, descent, and landing architecture studies [24, 25] and used in previous multi-fidelity modeling studies [12, 13]. For this study, both vehicles have a base radius of 9.4 m, shoulder radius 0.47 m, and half-cone and shoulder turn angles of 70° . A range of nose radii from 4 to 20 m will be examined in this study.

A. HIAD Specific Vehicle Parameters

Due to the use of the inflatable toroids, the HIAD vehicle is an axisymmetric, sphere-cone design. However, during entry, aerodynamic forces on the vehicle may cause the flexible TPS material to deform, pushing it into the region between toroids. This deformation of the TPS is referred to as surface scalloping and applies only to the HIAD vehicle in this study. This surface scalloping has been shown to augment the surface heat flux [7]. Thus, the scalloping surface deformation must be parameterized when defining the HIAD vehicle shape.

The surface scalloping is parameterized using the same technique described by Hollis [7], which was used as a

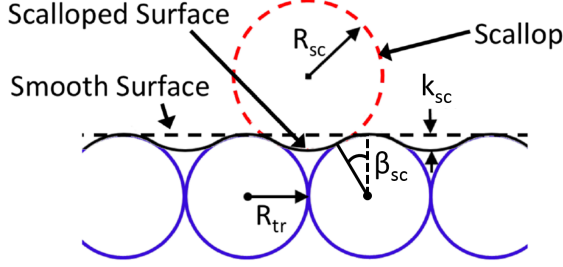


Fig. 6 Surface scalloping parameters.

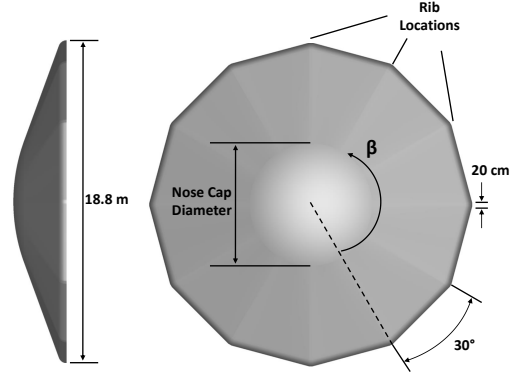


Fig. 7 Example ADEPT vehicle used in this study.

spherical to conical geometry transition location. Thus, the diameter of the solid nose cap changes proportionally with the nose radius. For this study, it is also assumed that there is a smooth transition from the solid nose cap to the flexible TPS region.

III. Trajectory Model

To model the entry, descent, and landing trajectories of the HIAD and ADEPT vehicles, a six degree-of-freedom (DOF) entry dynamics model was used. This six DOF entry model is the same used in a previous UQ analysis of a HIAD vehicle by Brune et al.[8]. The six equations of motion, given below in Eqs. 1 to 6, are used to determine the velocity, the position, and the flight direction of the vehicles. Where h is the altitude, $\hat{\theta}$ is the longitude, $\hat{\phi}$ is the latitude, $\hat{\gamma}$ is the flight path angle, and $\hat{\psi}$ is the heading.

In this study, the atmospheric density and temperature are modeled with NASA's MarsGRAM 2010 atmosphere model[27] including the built in random perturbations of the freestream density. The atmospheric interface is assumed to be at approximately 104 km above the reference Mars orbiter laser altimeter (MOLA) aeroid[27], and the vehicles are assumed to enter the atmosphere at 0 deg longitude and 0 deg latitude at an easterly heading of 90 deg. The state variables are obtained using numerical integration with a time step of 2 ms.

$$\dot{r} = V \sin \hat{\gamma} \quad (1)$$

$$\dot{\hat{\theta}} = \frac{V \cos \hat{\gamma} \cos \hat{\psi}}{r \cos \hat{\phi}} \quad (2)$$

$$\dot{\hat{\phi}} = \frac{V \cos \hat{\gamma} \sin \hat{\psi}}{r} \quad (3)$$

$$\dot{V} = \frac{D}{m} - g \sin \hat{\gamma} + \omega^2 r \cos \hat{\phi} (\sin \hat{\gamma} \cos \hat{\phi} - \cos \hat{\gamma} \sin \hat{\phi} \sin \hat{\psi}) \quad (4)$$

$$\dot{\hat{\gamma}} = \frac{L \cos \sigma}{m V} - \frac{g \cos \hat{\gamma}}{V} + \frac{V}{r} \cos \hat{\gamma} + \frac{(2\omega V)}{V} \cos \hat{\phi} \cos \hat{\psi} + \frac{\omega^2 r}{V} \cos \hat{\phi} (\cos \hat{\gamma} \cos \hat{\phi} + \sin \hat{\gamma} \sin \hat{\phi} \sin \hat{\psi}) \quad (5)$$

$$\dot{\hat{\psi}} = \frac{L \sin \sigma}{m V \cos \hat{\gamma}} - \frac{V}{r} \cos \hat{\gamma} \cos \hat{\psi} \tan \hat{\phi} + \frac{(2\omega V)}{V} (\tan \hat{\gamma} \cos \hat{\phi} \sin \hat{\psi} - \sin \hat{\phi}) + \frac{\omega^2 r}{V \cos \hat{\gamma}} \sin \hat{\phi} \cos \hat{\phi} \cos \hat{\psi} \quad (6)$$

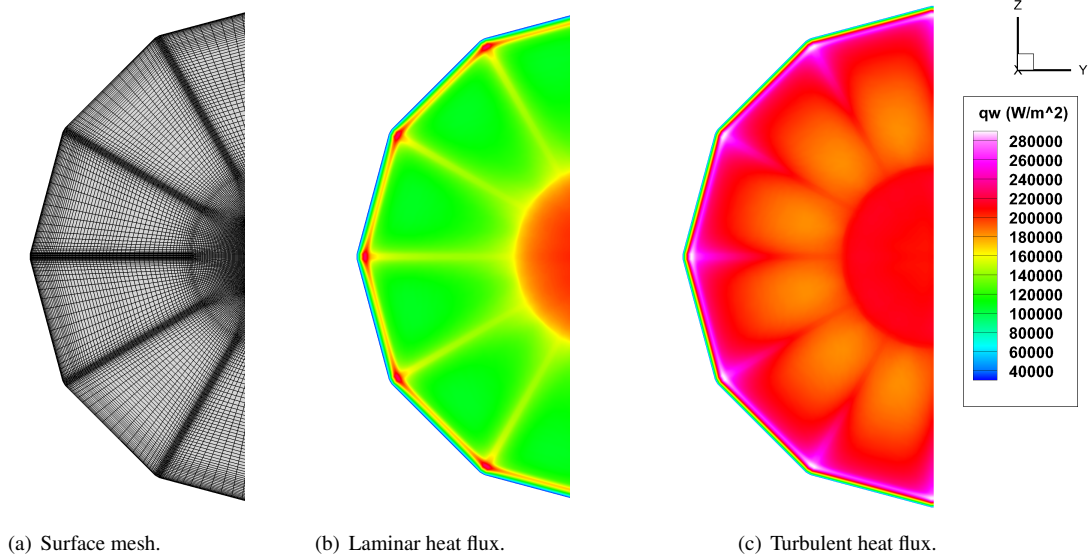


Fig. 8 Example ADEPT surface mesh and corresponding convective heat flux.

IV. Heating Models

A. LAURA CFD Predictions

The high-fidelity model used in this study is NASA Langley Research Center's high-speed thermochemical non-equilibrium Langley Upwind Relaxation Algorithm (LAURA) CFD software. This software is used as the high-fidelity computational model for both the laminar and turbulent heating solutions. The LAURA software utilizes an upwind relaxation algorithm and includes grid adaptation and interior grid clustering to effectively capture and model shock waves in the flow field[28] and to cluster the grid cells near the wall to obtain y^+ values of approximately one. The flow field is assumed to be steady state with a two temperature, thermochemical nonequilibrium gas model. The turbulence model used is the Cebeci-Smith algebraic model[29]. The Martian atmosphere is assumed to be composed of 97.417% CO_2 and 2.583% N_2 by mass with the flow field modeled by a 10 species composition model: CO_2 , CO , N_2 , O_2 , NO , C , N , O , CN , and C_2 . The reaction finite rate chemistry model that was used is given by Johnston and Brandis[30]. This model was tuned to match CO 4th Positive and CN Violet emission measured in the NASA Ames Research Center Electric Arc Shock Tube experiments. Flow field radiation was modeled by the High-Temperature Aerothermodynamic Radiation (HARA) code[31, 32] and was loosely coupled to the flow field solution using a tangent slab approximation to calculate the radiative heat flux and divergence of the radiative heat flux.

This study assumes that the vehicles are at zero degrees angle-of-attack (i.e., ballistic entry). Therefore, an axisymmetric grid is used for the HIAD vehicle. The HIAD axisymmetric grids have 96 cells in the surface-normal direction and 64 cells in the surface-parallel direction, as used in previous studies[12, 13]. For the ADEPT grids, half-geometry structured grids with an O-H grid topology for the vehicle surface are used. Each grid has approximately 1.84×10^6 cells, with 96 cells in the surface-normal direction. This grid size was shown to be sufficient in a previous study by Santos et al.[14]. A cold wall boundary condition was used for the vehicle surface along with a super-catalytic surface chemistry model and an absorptivity coefficient equal to one. These wall boundary conditions were chosen as they give a conservative prediction for convective heating. The cold wall boundary condition also allows for a hot wall correction to be applied to obtain corrected convective heat fluxes for higher wall temperatures. An example ADEPT surface grid used in this study and corresponding laminar and turbulent heat flux distributions are shown in Fig. 8. A solution was considered converged if the L2 norm was below 10^{-10} and if there was minimal change in the convective heating over the previous 100,000 iterations.

B. Sutton-Graves-Krasnov Correlations

The low-fidelity computational model used in this study is the Sutton-Graves with Krasnov laminar convective heating correlations. The function of this model is to provide a fast, correlation based prediction of the center line

laminar convective heat flux on an arbitrary axisymmetric blunt body at given flight conditions. The surface heat flux prediction starts with the calculation of the stagnation point heat flux with the Sutton-Graves correlation given by Eq. 7.

$$\dot{q}_0 = 1.83 \times 10^{-4} \sqrt{\rho/R_n} V^3 \quad [W/m^2] \quad (7)$$

Here, the stagnation point heat flux, $\dot{q}_0$, is a function of the freestream density, ρ , the vehicle nose radius, R_n , and the freestream velocity, V . For heating downstream of the stagnation point, correlations for the ratio of heating at an arbitrary point to the stagnation point from Krasnov[21] are used. The body is separated into its sphere and cone regions where different correlations for the heating ratio, $\frac{\dot{q}}{\dot{q}_0}$, are used for each region. For the spherical region, the following is used in terms of freestream Mach number, M_∞ ,

$$\frac{\dot{q}}{\dot{q}_0} = \frac{(1 - \zeta \sin^2 \phi) \phi \sin \phi}{[f_1(\phi) - \zeta f_2(\phi)]^{1/2}}. \quad (8)$$

where,

$$\zeta = 1 - \frac{1}{\gamma M_\infty^2}, \quad (9)$$

$$f_1(\phi) = \phi^2 - \phi \sin 2\phi + \sin^2 \phi, \quad (10)$$

$$f_2(\phi) = \frac{1}{2} \left[\frac{3}{2} \phi^2 + \phi \sin 2\phi \left(\frac{3}{2} + \sin^2 \phi \right) + \frac{3}{2} \sin^2 \phi + \frac{1}{2} \sin^4 \phi \right]. \quad (11)$$

Here, a frozen freestream specific heat ratio, γ , is assumed, and the surface inclination angle, ϕ , is measured from the normal direction at the nose to a point on the spherical region, as shown in Fig. 5. Next, the heating ratio on the conical surface is modeled by,

$$\frac{\dot{q}}{\dot{q}_0} = \frac{A \bar{\ell}}{\sqrt{B + \bar{\ell}^3}}. \quad (12)$$

where,

$$A(\delta) = \frac{\sqrt{3}}{2} \left\{ \left[\left(1 - \frac{1}{\gamma M_\infty^2} \right) \sin^2 \delta + \frac{1}{\gamma M_\infty^2} \right] \left(\frac{\pi}{2} - \delta \right) \right\}^{\frac{1}{2}}, \quad (13)$$

$$B(\delta) = \frac{\frac{3}{16} D(\phi) / \phi}{\left(1 - \frac{1}{\gamma M_\infty^2} \right) \sin^4 \delta + \frac{\sin^2 \delta}{\gamma M_\infty^2}} - \cot^3 \delta, \quad (14)$$

$$D(\phi) = \left(1 - \frac{1}{\gamma} \right) \left(\phi^2 - \frac{\phi}{2} \sin(4\phi) + \frac{1}{8} (1 - \cos(4\phi)) \right) + \frac{4}{\gamma} \left(\phi^2 - \phi \sin(2\phi) + \frac{1}{2} (1 - \cos(2\phi)) \right) \quad (15)$$

$$\bar{\ell} = \frac{\ell + s}{R_n}. \quad (16)$$

To ensure a continuous heat flux distribution, an offset to the running length, ℓ , is added to the heat flux in the conical region. This offset, s , is found by equating the heat flux found in both regions and solving for the offset using the secant method.

C. West-Brandis Mars Radiative Heating Correlation

The low-fidelity model used for the prediction of radiative heat flux utilizes the updated Mars heating polynomial correlations by West and Brandis[33] for the stagnation point radiative heat flux prediction. This correlation is given by Eq. 17. In this correlation, $f(V, \rho, R_n)$ is a fourth order polynomial and is a function of freestream velocity, freestream density, and effective nose radius.

$$\dot{q}_r = e^{f(V, \rho, R_n)} \quad (17)$$

Note that while this correlation is a function of effective nose radius and assumes an adiabatic wall temperature, this study uses the physical nose radius and assumes a cold wall temperature. Another important aspect of this correlation to note is that this low-fidelity model is separated into two regions, with each region having different set of coefficients: a low velocity region for velocities between 2 and 6 km/s and a high velocity region for velocities between 6 and 8 km/s. The reason for this separation is because, as shown by West and Brandis, the radiative heat flux increases dramatically as the velocity increases to 6 km/s and beyond due to the activation of the CO 4th Positive and the CN Violet band systems.

In the off-stagnation region, no engineering correlations exist for prediction of the radiative heat flux distribution. Therefore, the radiative heat flux along the body was assumed to be linear with a slope equal to the negative inverse of the stagnation point value. This distribution was chosen because the co-Kriging method requires that the response variable varies with at least one of the independent variables. Choosing this distribution as the low-fidelity model ensures that the off-stagnation radiative heat fluxes vary over the sample space.

D. Multi-Fidelity Model

Multi-fidelity models are low cost surrogate models of high cost, high-fidelity models that use a low-fidelity model to reduce the number of high-fidelity model evaluations used to construct the surrogate model. In this study, multi-fidelity co-Kriging models are used as a surrogate models for the high-fidelity LAURA CFD heat flux predictions. Co-Kriging is an interpolative method that has also been used extensively in low-speed aerospace applications[15, 17]. For this study, a CKMF surrogate modeling approach was used that was developed previously by Santos et al.[12–14] for the prediction of the convective and radiative heat flux on HIAD and ADEPT vehicles, including the prediction of the scalloped augmented convective heat flux on HIAD vehicles. In these previous studies, the co-Kriging based multi-fidelity models were shown to have a mean accuracy within 10% when compared to high-fidelity LAURA CFD solutions. This approach uses the LAURA CFD predictions as the high-fidelity model, and the Sutton-Graves-Krasnov and West-Brandis correlations as the low-fidelity models. This approach also includes a distance-weighted, root-mean-squared error (RMSE) adaptive sampling approach and parameterization of the low and high-fidelity heat flux distributions using Hicks-Henne bump functions.

1. Co-Kriging Approach

A multi-fidelity co-Kriging model is built off a Kriging surrogate model of the low-fidelity model. A Kriging model is a surrogate model that interpolates an unknown function, $f(\mathbf{x})$, using a Gaussian process. This model assumes that the unknown function response, $\hat{y}$, at any location $\mathbf{x}$ in a k sample space is equal to a global mean, μ , plus some random variation $w\Psi$, as shown in Eq. 18.

$$\hat{y}(\mathbf{x}) = \mu + w\Psi \quad (18)$$

A co-Kriging model, $\hat{y}_{MF}$, is then simply a difference model between the high-fidelity model, y_{HF} , and a Kriging surrogate of the low-fidelity model, $\hat{y}_{LF}$, as shown in Eq. 19, where ν is a weighting value.

$$\hat{y}_{MF} = y_{HF} - \nu\hat{y}_{LF} \quad (19)$$

Although a simple concept, the process of building a co-Kriging model from observed data is quite involved and requires the optimization of several hyper-parameters using global search methods to fit the model to the data. For a more detailed treatment of co-Kriging, see Forrester et al.[34].

In this study, the CKMF models were built as functions of freestream density, velocity, and vehicle nose radius. These are the same independent variables used in the low-fidelity models. The sample space used in this study is the same used in previous studies by Santos et al.[12–14] and is given in Table 1. This sample space was chosen because it

Table 1 The boundaries of the three-dimensional sample space used in analysis.

	Nose Radius [m]	Flight Velocity [km/s]	Density [kg/m ³]
Minimum	4	2	6.6752x10 ⁻⁵
Maximum	20	7	1x10 ⁻³

was shown to contain the maximum stagnation point convective heat flux trajectory points for a wide range of possible entry velocities and vehicle masses.

In this study, the low-fidelity model was sampled linearly in each dimension, and a distance-weighted RMSE adaptive sampling plan was used for the high-fidelity model. Along with the function response, the co-Kriging model can also estimate the RMSE of the prediction at any point. In the distance-weighted RMSE sampling scheme, at each iteration of the adaptive sampling process, the RMSE of the stagnation point prediction is calculated across the entire sample space. The RMSE at every location is then weighted by the distance to the nearest training point. New training points are then added at the locations with the largest weighted RMSE values, and the process is repeated with the updated training set. Using this process, previous studies by Santos et al. have shown that, along with using 512 low-fidelity training sample points, 36 and 48 high-fidelity training samples produce converged CKMF models for convective (both laminar and turbulent) and radiative heating, respectively.

A previous study by Santos et al. [12] demonstrated that applying the CKMF modeling technique to Hicks-Henne parameterized [35] output distributions instead of each surface point individually resulted in predictions that were more computationally efficient and accurate. This study also uses the Hicks-Henne parameterization technique to parameterize the surface heat flux distributions. The Hicks-Henne shape parameterization technique is commonly used in aerospace engineering applications for parameterization of airfoil shapes [35]. In this technique, a set of n control points on a baseline two-dimensional shape are chosen. A linear combination of sine functions over the n control points, shown in Eq. 20, are used to perturb the baseline shape, $y(x)^i$, to get the desired “deformed” shape, $y(x)$. Here, bump amplitude coefficients, a_j , bump width coefficients, b_j , and the x -locations of the control points, c_j , are used to define the shape deformation. Thus to model the heat flux distributions, each amplitude coefficient is modeled by a separate CKMF model over the three-dimensional sample space. In this study, the baseline shape was assumed to be a straight line at a constant value of one, and the heat flux distributions are scaled by the stagnation point value. The 22 control point locations used in the previous studies by Santos et al. [12, 13] are also used in this study.

$$y(x) = y^i(x) + \sum_{j=0}^n a_j \sin^{b_j}(\pi x^{\ln(0.5/c_j)}) \quad (20)$$

2. Application to HIAD Vehicle

To model the augmented turbulent heating on HIAD vehicles due to surface scalloping, the correlation developed by Hollis [7] is used. This correlation uses a combination of laminar and turbulent convective heat flux solutions to predict the augmented heat flux caused by surface scalloping. In terms of the CKMF modeling process, this means that CKMF models of the laminar and turbulent heat fluxes must first be obtained. The Hollis correlation is then subsequently applied to these CKMF models to obtain a model of the scalloped augmented heat flux. This correlation is a function of scallop depth, k_{sc} , and is based on a linear combination of the laminar and turbulent solutions. The correlation is given by the form shown in Eq. 21.

$$h_{k,turb} = (A_k)(h_{lam}) + (B_k/B_0)(\delta h_{turb}) \quad (21)$$

Here the scalloped augmented heat transfer coefficient, $h_{k,turb}$, is a function of a constant, B_0 , two scallop depth dependent constants, A_k and B_k , the laminar solution heat transfer coefficient, h_{lam} , and the difference between the turbulent and laminar solution heat transfer coefficients, δh_{turb} . Note that, although the wind tunnel experimental data shows that scalloping causes peaks and valleys in the surface convective heat flux, this correlation creates a smooth heat flux distribution that puts an upper bound to the peak values. Also note that in the nose cap region, before the flow has transitioned to turbulent, this correlation over-predicts because it assumes fully turbulent flow everywhere. For more details on this correlation and its use in multi-fidelity modeling, see studies by Hollis[7] and Santos et al.[13]. Co-Kriging based multi-fidelity heat flux results for a selected HIAD test case are plotted in Fig. 9 along with the high-fidelity LAURA CFD results for comparison.

3. Application to ADEPT Vehicle

As previously stated, the ADEPT vehicle is non-axisymmetric which leads to a surface heat flux distribution that is also non-axisymmetric. Therefore, the described CKMF process needs to be applied to a two-dimensional heat flux distribution. For this study, the ADEPT geometries and heat flux distributions have a plane of symmetry every 15 degrees in the azimuthal direction starting at a rib center line. Therefore, only a 15 degree section of the vehicle need

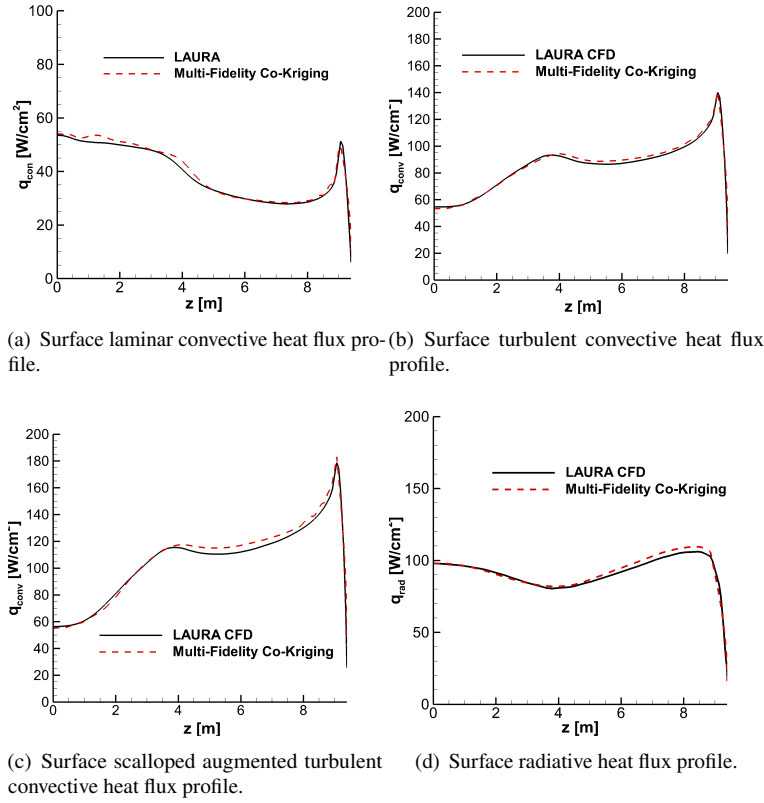


Fig. 9 HIAD CKMF model surface heat flux compared to high-fidelity LAURA CFD heat flux for a selected test case (velocity = 7.00 km/s, density = 5.82×10^{-4} kg/m³, nose radius = 12.22 m).

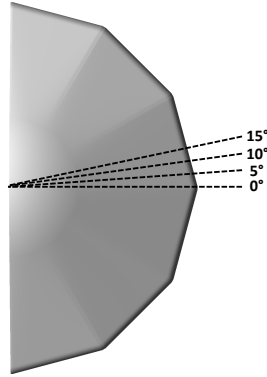
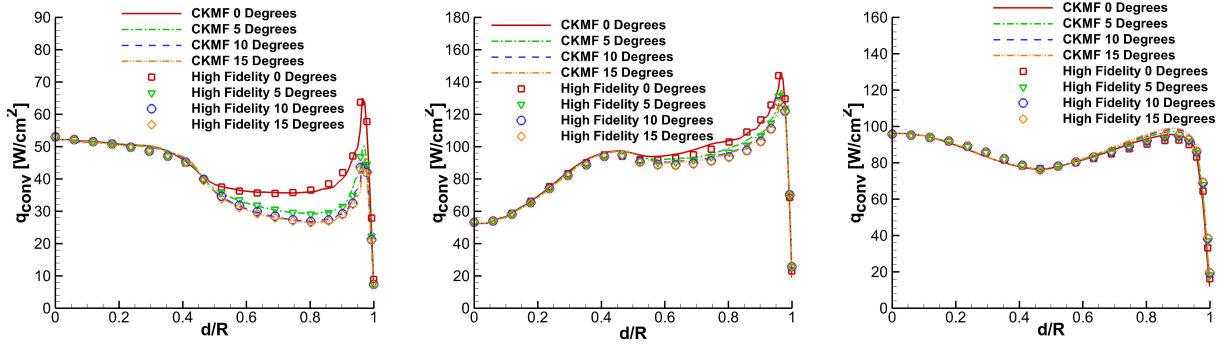


Fig. 10 Solution slice locations for ADEPT CKMF process.

actually be modeled. Thus, to apply the described CKMF approach to the ADEPT, each geometry is evaluated along four lines across the surface: 0 deg, 5 deg, 10 deg, and 15 deg, as shown in Fig. 10. The low-fidelity model is evaluated along these lines using the surface inclination, and the high-fidelity LAURA CFD results are sliced along these lines. The described CKMF approach with Hicks-Henne parameterization is then applied to each line separately. Co-Kriging based multi-fidelity heat flux results for a selected ADEPT test case are plotted in Fig. 11 along with the high-fidelity LAURA CFD results for comparison.



(a) Surface laminar convective heat flux profile. (b) Surface turbulent convective heat flux profile. (c) Surface radiative heat flux profile.

Fig. 11 ADEPT CKMF model surface heat flux compared to high-fidelity LAURA CFD heat flux for a selected test case (velocity = 7.00 km/s, density = 5.82×10^{-4} kg/m³, nose radius = 12.22 m).

4. CKMF Model Prediction Error

While the CKMF approach results in high accuracy surrogate models, these models do have some error in their predictions. These errors must be estimated and modeled to capture their effects on heating uncertainty predictions. For this study, 36 test points chosen using a random Latin hypercube sampling scheme were used to estimate the CKMF model error. The CKMF model and the high-fidelity model were then both evaluated at the test points. The error of the CKMF model was then calculated by integrating the absolute difference between the CKMF and high-fidelity models along each surface line as shown in the equation below:

$$\dot{q}_{Err} = \frac{1}{R_b} \int_0^{R_b} |\dot{q}_{MF} - \dot{q}_{HF}| dz \quad (22)$$

Here $\dot{q}_{Err}$ is the CKMF heat flux model error, $\dot{q}_{MF}$ is the CKMF model heat flux prediction, and $\dot{q}_{HF}$ is the high-fidelity model heat flux prediction. A second order polynomial was then fitted to the error at the 36 test points to obtain a model of the estimated CKMF heat flux error as a function of freestream density, velocity, and nose radius. The R-squared values of the polynomial fits were maximized by eliminating extraneous terms using coefficient p-values. In the UQ analyses, the CKMF error models are considered to be an epistemic error.

E. Hot Wall Heating Correction

The heating models described in the previous sections all assume a cold wall surface temperature which means that they may over predict the surface heat fluxes if the wall temperatures increase throughout the trajectory. Thus, to get a more realistic heating prediction, changing wall temperatures over the trajectory must be taken into account. In this study, a simplified steady state heat conduction model is used to calculate the TPS wall temperatures at each time step. This simplified heating model is given in Eqs. 23 and 24 below and is illustrated in Fig.12.

$$\dot{q}_c + \dot{q}_r - \epsilon \sigma T_w^4 = \dot{q}_{tot} \quad (23)$$

$$\dot{q}_{tot} = \frac{k_1}{L_1} (T_w - T_1) = \frac{k_2}{L_2} (T_1 - T_2) = \frac{k_3}{L_3} (T_2 - T_e) \quad (24)$$

Here $\dot{q}_c$ is the hot wall convective heat flux to the surface, $\dot{q}_r$ is the radiative heat flux to the surface, $\dot{q}_{tot}$ is the total heat flux conducted through the TPS, σ is the Stefan-Boltzmann constant, ϵ is the surface emissivity, $k_{1,2,3}$ are the thermal conductivities of each TPS layer, $L_{1,2,3}$ are the thicknesses of each TPS layer, T_w is the TPS surface temperature, T_e is the temperature on the backside of the TPS, and $T_{1,2}$ are the temperatures at the interface of the TPS insulating layer. As mentioned in section IV.A, the absorptivity in this model is assumed to be equal to one. In this study, this steady state heat conduction model is simplified further by assuming that the temperature at the back side of the TPS, T_e , is equal to the freestream temperature. Thus, the temperatures at T_w , T_1 , and T_2 can be found given a net surface heat flux $\dot{q}_{tot}$, a freestream temperature, and TPS material properties.

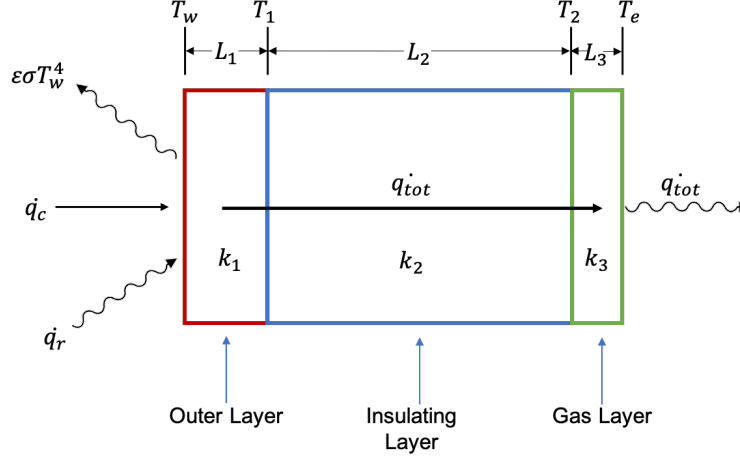


Fig. 12 Diagram of TPS layers and steady state heat fluxes.

A study by Brune et al.[10] has shown that the corrected hot wall convective heat flux can be calculated using the following equation:

$$\dot{q}_c = h(H_{0e} - H_w) \quad (25)$$

Where h is the heat transfer coefficient, H_{0e} is the total boundary layer edge enthalpy and H_w is the wall enthalpy. In this study, because the total enthalpy across a shock does not change, it is assumed that the boundary layer edge enthalpy equals the freestream enthalpy. The study by Brune et al. also demonstrated that for Mars entry, h can be assumed to be independent of wall temperature. Thus, knowing the cold wall convective heat fluxes, the corrected hot wall convective heat fluxes during Mars atmospheric entry can be found assuming the heat transfer coefficient is independent of wall temperature. Note that Eqs. 23, 24, and 25 are all functions of the wall temperature. The wall temperature and hot wall convective heat flux are solved for simultaneously by substituting Eq. 25 into Eq. 23 and then solving Eqs. 23 and 24 using an iterative process with Newton's method.

It is important to note that in this study, it is assumed that the both the HIAD and ADEPT use the same generic TPS material and properties. However, in reality, the TPS materials and properties for ADEPT vehicles will be different than that of HIAD vehicles. It is also assumed that the TPS is the same material and has the same thickness over the entire vehicle. These two assumptions allow for relative comparison of ADEPT and HIAD heating based solely on vehicle geometry, eliminating differences in heating due to the use of different TPS materials. For this study, the TPS is assumed to be made up of three layers: an outer layer, a insulating layer, and a gas layer as illustrated in Fig. 12. The material properties and thicknesses used for each layer are based on TPS materials used in the development of the HIAD vehicle. Finally, note that the radiative heat loads presented in the results include both the radiative heat flux to and away from the surface.

V. Results

First, the use of CKMF heat flux models in a trajectory Monte Carlo analysis is demonstrated and verified assuming fully laminar flow for a HIAD vehicle. The results using the CKMF model are compared to results using the low-fidelity Sutton-Graves model and results using a single-fidelity Kriging surrogate of the High-fidelity model to assess model validity. Next, the CKMF heating models are used in Monte Carlo uncertainty analyses of heat load distributions on HIAD and ADEPT vehicles over a range of vehicle nose radii from four to twenty meters.

A. Description of Stochastic Problem

As previously stated, the vehicles analyzed in this study have a base radius of 9.4 meters, a half-cone angle of 70 degrees, and a shoulder turn radius of 0.47 m. In this study, the trajectory model variables with epistemic uncertainties are assumed to be the entry interface altitude, h_{EI} , and the drag coefficient, C_D . The trajectory variables with aleatory uncertainties are assumed to be the entry velocity, V_{EI} , the entry flight path angle, γ_{EI} , and the vehicle mass, m . The

MarsGram 2010 atmosphere model used in the trajectory calculations also contains uncertainties in the freestream conditions. Finally, the errors in the CKMF heating models, $\dot{q}_{Err}$, are assumed to be epistemic uncertainties. Note that this study focuses on uncertainties in the trajectory and CKMF models, with uncertainties in the CFD model detailed by West et al.[36] and Brune et al.[10] omitted from consideration. A more detailed UQ study can be performed in the future by including the omitted sources of uncertainty. The nominal case is a 7200 kg vehicle that has a drag coefficient of 1.5, an entry velocity of 7300 m/s, an entry altitude of 104 km, and an entry flight path angle of -15 deg. Table 2 summarizes the uncertain variables used in this study along with their type and distributions (mean and standard deviation, or minimum and maximum). The trajectory and atmospheric variables and distributions used are derived from previous Mars deployable entry vehicle EDL trajectory and heating UQ studies by Cianciolo et al.[25] and Brune et al.[8].

Table 2 Uncertain variable information for Mars EDL problem.

Variable	Distribution	Mean/Min.	Std./Max
h_{EI} , km	Epistemic	103.9	104.1
C_D	Epistemic	1.35	1.65
$\dot{q}_{Err}$, W/cm ²	Epistemic	$-f(\rho, V, R_n)$	$+f(\rho, V, R_n)$
V_{EI} , m/s	Gaussian	7300	7.3
γ_{EI} , deg	Uniform	-15.1	-14.9
m , kg	Gaussian	7200	167
Freestream Density, kg/m ³	Mars GRAM Model	Mars GRAM Model	Mars GRAM Model
Freestream Temperature, K	Mars GRAM Model	Mars GRAM Model	Mars GRAM Model

B. Monte Carlo with CKMF verification

The Monte Carlo with CKMF verification study is performed using a Monte Carlo analysis with a Latin hypercube sampling scheme on only the four meter nose laminar HIAD geometry. The Latin hypercube sampling scheme was chosen to help speed up convergence of the Monte Carlo analysis. Only the four meter nose radius HIAD vehicle is used in the verification study because it represents the case with highest stagnation point heat fluxes and is the simplest geometry, which lowers computational cost.

The first step in the analysis process is to determine the Monte Carlo sample size needed to obtain converged results. The Monte Carlo simulations were run along the trajectory using the CKMF heating prediction with 4,000, 10,000, 50,000, and 100,000 samples for a four meter nose radius vehicle. The resulting 95% confidence intervals of the convective heat flux distributions at maximum stagnation point heating are shown in Fig.13 and indicate that 10,000

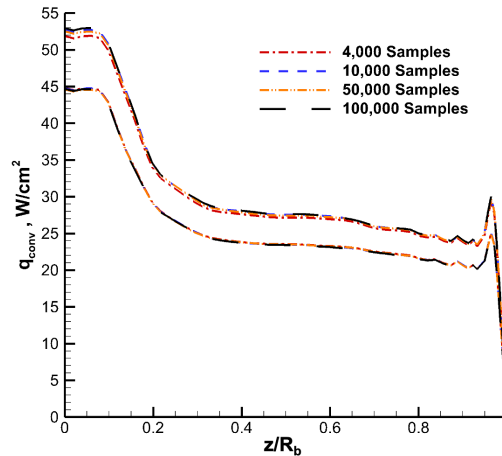


Fig. 13 Convective heat flux 95% confidence interval convergence.

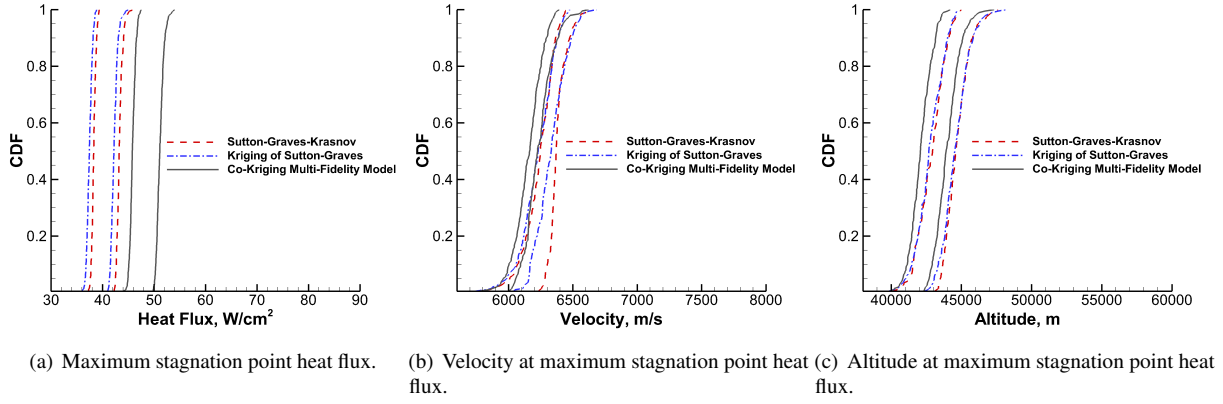


Fig. 14 P-box results for maximum cold wall stagnation point convective heat flux.

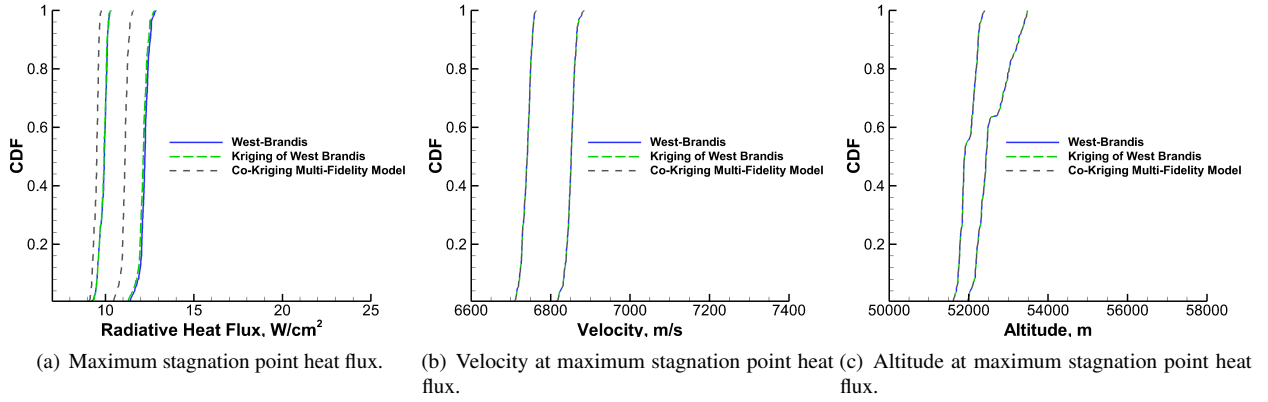


Fig. 15 P-box results for maximum cold wall stagnation point radiative heat flux.

Monte Carlo samples are sufficient to obtain converged results. Using 10,000 samples, the maximum stagnation point heat fluxes, the velocities at maximum stagnation heating, and the altitudes at maximum heating were calculated for convective and radiative heating. The probability box (P-box) results for convective and radiative heating are shown in Figs. 14 and 15, respectively. Figure 14 also contains convective heating results using the Sutton-Graves-Krasnov model and the single-fidelity Kriging surrogate of the Sutton-Graves-Krasnov model. Figure 15 contains the radiative heating results using the West-Brandis model and the single-fidelity Kriging surrogate of the West-Brandis model. Note that only the cold wall heat fluxes are used in the verification study to allow for comparison between model fidelities by eliminating wall temperature effects.

From Fig. 14, it can be seen that both the Sutton-Graves-Krasnov and the single-fidelity Kriging heating models have very similar results, which is expected because the Kriging model is a surrogate of the Sutton-Graves-Krasnov. This similarity also indicates that the single-fidelity Kriging model, and thus the multi-fidelity co-Kriging model, are implemented correctly in the trajectory model, and that there are no anomalies in either surrogate model. Figure 14 also shows that the maximum heat flux prediction from the multi-fidelity and low-fidelity heating models do not overlap, and the low-fidelity model completely under predicts the maximum heat flux. In Fig. 15, the velocity and altitude P-boxes are exactly the same for all three models indicating that the maximum radiative heat flux occurs on the edge of the CKMF model computational domain. The maximum radiative heating occurring at the edge of the CKMF model domain is expected because radiative heating increases exponentially with velocity at velocity ranges above 6 km/s. Figure 15(a) shows that while the multi-fidelity and low-fidelity models have similar heating predictions, the multi-fidelity model has smaller confidence interval than the low-fidelity model.

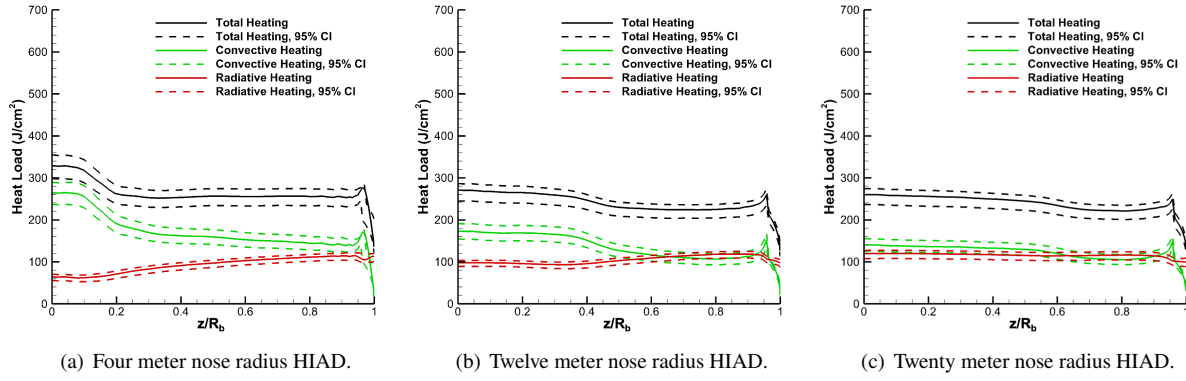


Fig. 16 Nominal laminar flow heat loads with 95% confidence intervals for a four meter, a twelve meter, and a twenty meter nose radius HIAD vehicle.

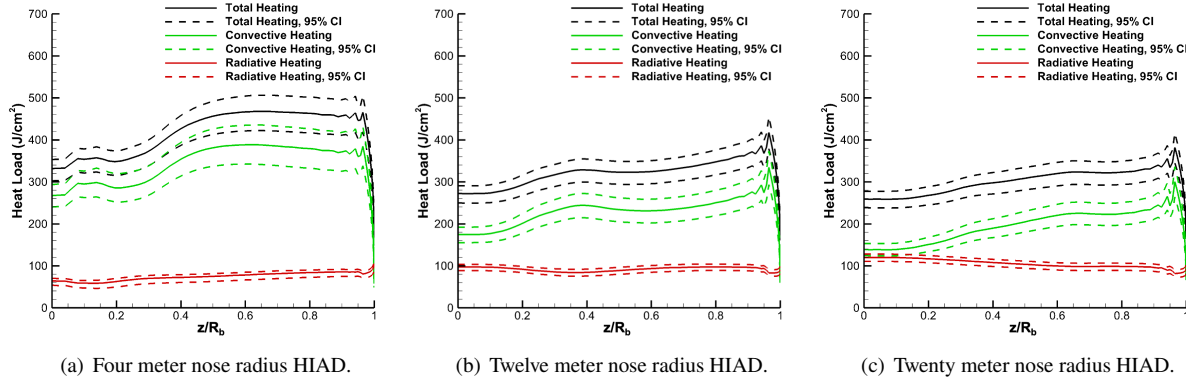


Fig. 17 Nominal turbulent flow heat loads with 95% confidence intervals for a four meter, a twelve meter, and a twenty meter nose radius HIAD vehicle.

C. HIAD Heat Load Study

The Monte Carlo approach was used to obtain 95% confidence intervals of the surface heat load distributions on HIAD vehicles with the hot wall heating correction and nose radii of: 4 m, 6 m, 8 m, 10 m, 12 m, 14 m, 16 m, 18 m, and 20 m. The heat load distributions are obtained by numerically integrating the surface heat flux distributions over flight time as shown in Eq. 26.

$$Q = \int_{t_0}^{t_f} \dot{q} dt \quad [J/cm^2] \quad (26)$$

The nominal surface heat load results with 95% confidence intervals for three selected vehicle geometries (four meter, twelve meter, and twenty meter nose radius) are shown in Figs. 16 and 17 for fully laminar and fully turbulent flow, respectively. These figures show results for convective and radiative heat load separately as well as the total (convective + radiative) heat load. For laminar flow, the maximum total heat load uncertainties and nominal total heat loads occur at the stagnation point for all three selected cases. For turbulent flow, the maximum total heat load uncertainty and nominal total heat load occur at the shoulder for all three selected cases. For all cases, the maximum total heat load uncertainty occurs at the surface location of the maximum total nominal heat load. The maximum nominal total heat load, maximum total heat load uncertainty, and maximum total heat load uncertainty as a percentage of the nominal value for the three selected HIAD cases are given in Table 3. Note that the uncertainty intervals are not centered on the nominal values and are biased slightly low. Figures 16 and 17 also show that for both the fully laminar and turbulent cases, the uncertainty in the convective heat load is the main contributor to the uncertainty in total heat load.

Table 3 Maximum nominal total heat loads, maximum total heat load 95% confidence interval, and maximum total heat load 95% confidence interval as a percentage of nominal value.

Nose Radius, m	Nominal Laminar, J/cm ²	Laminar Uncertainty, J/cm ²	Laminar % Uncertainty	Nominal Turbulent, J/cm ²	Turbulent Uncertainty, J/cm ²	Turbulent % Uncertainty
HIAD						
4	329.0	[298.5, 354.7]	[-9.27, 7.81]	465.8	[419.5, 505.3]	[-9.94, 8.48]
12	270.8	[245.1, 286.4]	[-9.49, 5.76]	417.8	[378.7, 454.7]	[-9.36, 8.83]
20	260.3	[236.8, 274.7]	[-9.03, 5.53]	382.1	[345.3, 416.3]	[-9.63, 8.95]
ADEPT						
4	332.2	[302.8, 356.2]	[-8.85, 7.22]	425.7	[386.0, 465.5]	[-9.33, 9.35]
12	304.0	[280.9, 323.9]	[-7.60, 6.55]	382.2	[345.6, 414.2]	[-9.58, 8.37]
20	282.2	[262.4, 300.9]	[-7.02, 6.21]	352.8	[321.5, 380.1]	[-8.87, 7.74]

The 95% confidence interval at the stagnation point ($z/R_b = 0$), the flank ($z/R_b = 0.5$), and the shoulder ($z/R_b = 0.964$) for all nine vehicle nose radii are shown in Figs. 18 and 19. The uncertainty trends displayed in Figs. 16 and 17 are again displayed in Figs. 18 and 19. Note that uncertainty at each surface point varies minimally across the entire range of nose radii and that the uncertainty increases only slightly with increasing heat load values. This means that the vehicle nose radius has a minimal effect on the heat load prediction uncertainties and that the error in the CKMF models likely has minimal effect on the heating prediction error as well. Finally, note that the flank and shoulder locations can experience as much or more heating and uncertainty as the stagnation point.

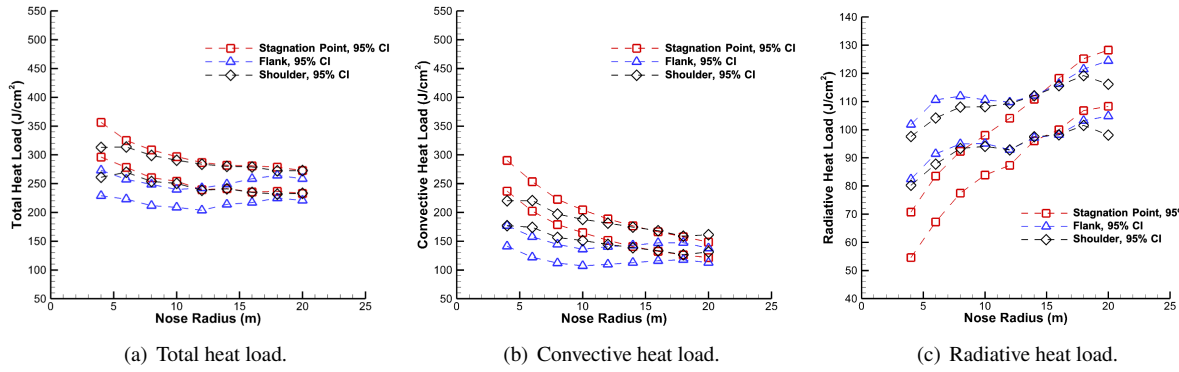


Fig. 18 Laminar heat load 95% confidence intervals vs. vehicle nose radius at the three select surface locations for HIAD vehicle.

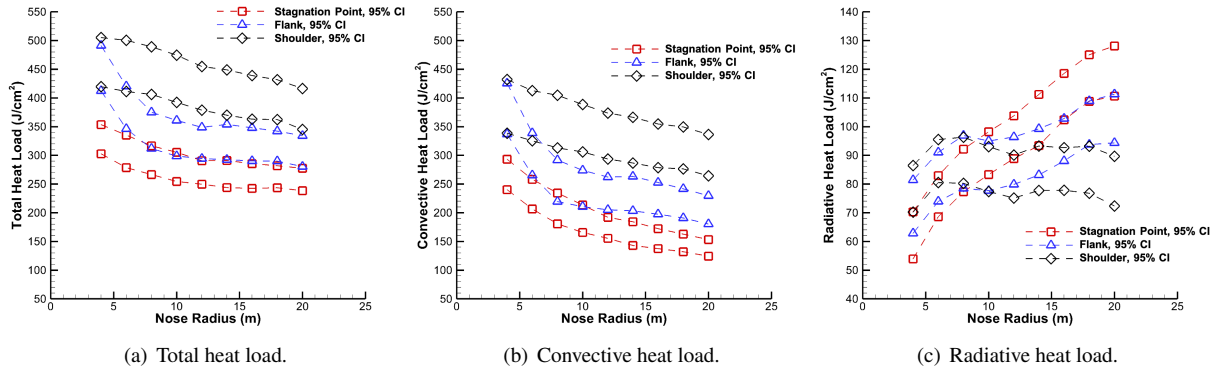


Fig. 19 Turbulent heat load 95% confidence intervals vs. vehicle nose radius at the three select surface locations for HIAD vehicle.

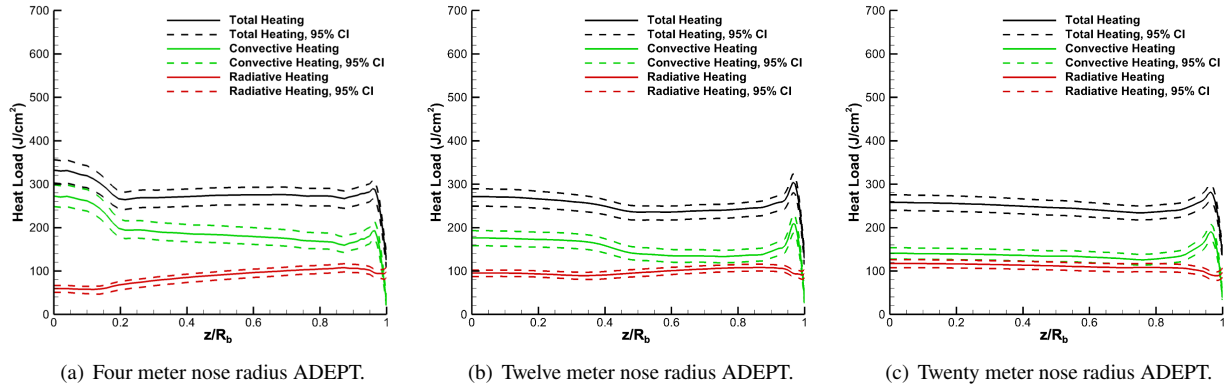


Fig. 20 Nominal laminar flow heat loads with 95% confidence intervals for a four meter, a twelve meter, and a twenty meter nose radius ADEPT vehicle.

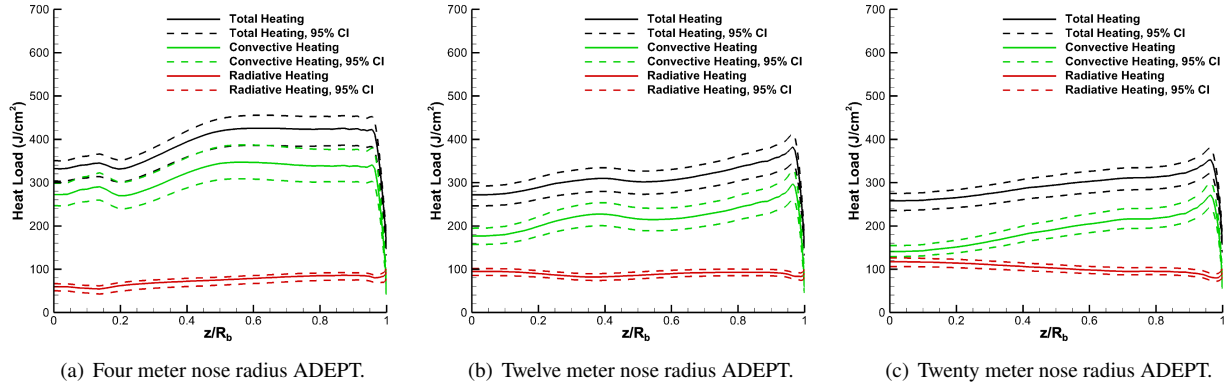


Fig. 21 Nominal turbulent flow heat loads with 95% confidence intervals for a four meter, a twelve meter, and a twenty meter nose radius ADEPT vehicle.

D. ADEPT Heat Load Study

The Monte Carlo process was repeated for the ADEPT vehicle during Mars atmospheric entry, again using the hot wall heating correction and the same nose radii used in the HIAD study. Note that the maximum heat loads and uncertainties for the ADEPT vehicle occur along the 0 degree azimuthal (rib) locations. Therefore, only the results for the 0 degree location are presented in this paper. The heat load distributions and 95% confidence intervals along the zero degree slice are shown in Figs. 20 and 21 for fully laminar and fully turbulent flows, respectively. Note that the results for the ADEPT vehicle are very similar to the results for the HIAD vehicle for both laminar and turbulent flow. Similar to the HIAD, convective heat load uncertainty for the ADEPT is the main contributor to the total heat load uncertainty, with this contribution reduced as the vehicle nose radius increases. Again, the maximum total heat load uncertainties occurs at the locations of the maximum nominal total heat loads. The maximum total heat load uncertainty and nominal total heat load for both the twelve and twenty meter cases occur at the shoulder, for both laminar and turbulent flow. For the four meter case, the maximum total heat load uncertainty and the maximum nominal total heat load occurs at the stagnation point for laminar flow and at the flank for turbulent flow. The maximum nominal total heat load, maximum total heat load uncertainty, and maximum total heat load uncertainty as a percentage of the nominal value for the three selected ADEPT cases are given in Table 3.

The 95% confidence interval at same three select surface locations (stagnation point, flank, and shoulder) for all nine vehicle nose radii are shown in Figs. 22 and 23. Again, the variation in uncertainty at each surface location across nose radii is minimal. This suggests that like for the HIAD vehicle, the nose radius has a minimal effect on the heating uncertainties and that CKMF model error is not a main contributor to heat load uncertainties. As with the HIAD, the

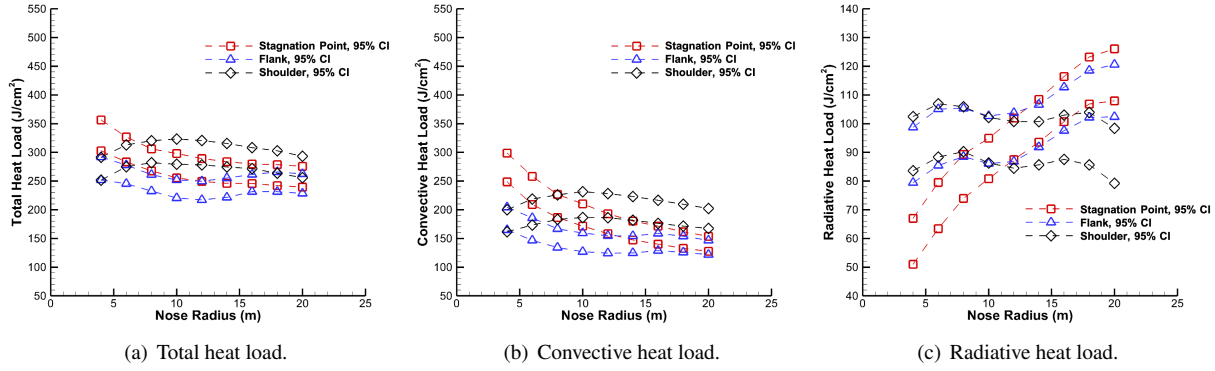


Fig. 22 Laminar heat load 95% confidence intervals vs. vehicle nose radius at the three select surface locations for ADEPT vehicle.

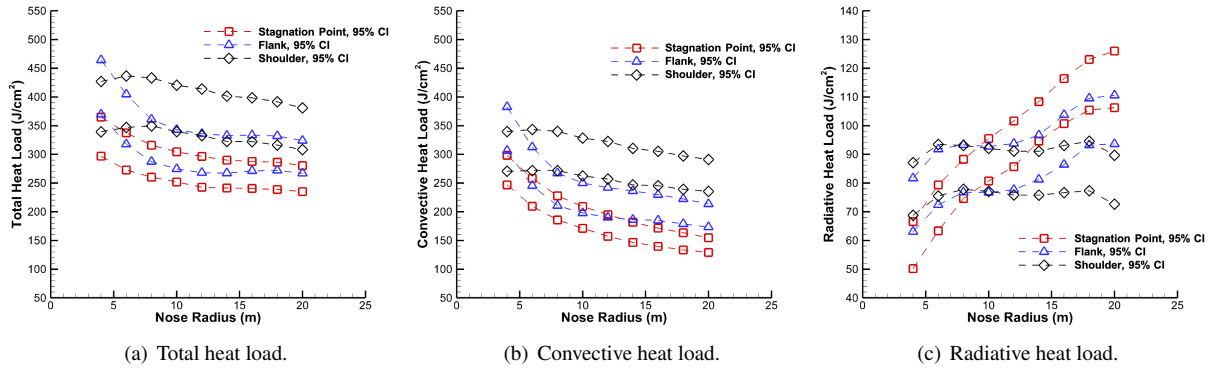


Fig. 23 Turbulent heat load 95% confidence intervals vs. vehicle nose radius at the three select surface locations for ADEPT vehicle.

flank and shoulder regions of the ADEPT can experience as much or more heating and uncertainty compared to the stagnation point.

Comparing the maximum uncertainty results for the HIAD to the ADEPT in Table 3 shows that the results are very similar, with only a 1 to 2 % difference in the uncertainty as a percentage of nominal heat load. Comparing the uncertainties along the vehicle surface and across nose radii between the ADEPT and the HIAD in Figs. 16 through 23 shows similar trends in the heat load uncertainties for both vehicles. As mentioned in Section IV.E, the use of the same generic TPS for both vehicles in this study means that a comparison in heating between the two vehicles is a relative comparison of geometry based heating. Therefore, there is almost no difference in geometry based heating and heating uncertainty between the two vehicles. This lack of difference in geometry based heating means that when designing and selecting HIAD or ADEPT vehicle technologies for missions, it is the vehicle TPS material that will determine any differences in heating between the two vehicles, not the vehicle geometries.

VI. Conclusion

The objective of this work was to demonstrate and verify the use of CKMF models in UQ analyses of surface heating on HIAD and ADEPT vehicles during Mars entry. A second objective of this work was to use the results from the UQ analyses to make a relative comparison of heating, based on vehicle geometry, between the HIAD and the ADEPT. In these UQ analyses, the CKMF models were used to predict the surface convective and radiative heat flux. Using a Monte Carlo analysis, the CKMF modeling approach was shown to produce accurate results for the maximum stagnation point heat flux when compared to other heating models. For the heat load analyses, the total heat load uncertainties

were found to vary minimally with vehicle nose radius for both the HIAD and ADEPT. The study also found that the maximum heat load uncertainties occurred at the maximum heat load locations for both vehicles as well. The error in the CKMF model was found to minimal effect on the vehicle heat load uncertainties.

The results of this study showed that there is very little difference in heating and heating uncertainties based on vehicle geometry between the HIAD and the ADEPT. This lack of difference is important because the two vehicles have very different surface geometries. Comparing the ADEPT (a non-axisymmetric vehicle that has cornered sections at the ribs and large flat sections in the regions between the ribs) with the HIAD (an axisymmetric vehicle with surface scalloping and no ribs), the expectation would be that these differences in surface geometry would have a significant impact on heating. However, the results of this study show that these significant differences in vehicle surface geometry do not have a significant impact on the vehicle heating. Therefore, in deciding between the HIAD or ADEPT during mission design, it is the TPS material and construction that will dictate any differences in heating between the two vehicles not the vehicle geometries.

VII. Acknowledgements

This work was supported by a NASA Space Technology Research Fellowship (NSTRF) under grant no. 80NSSC17K0170 (Mario Santos, NSTRF student, Serhat Hosder, principal investigator, and Thomas K. West, research collaborator). The first two authors would like to also acknowledge the use of Missouri S&T's high performance computing cluster *Foundry* for this research, which was developed under National Science Foundation Grant No. OAC-1919789.

References

- [1] Polsgrove, T., and Dwyer-Cianciolo, A. M., "Human Mars Entry, Descent and Landing Architecture Study Overview," *AIAA SPACE 2016*, Long Beach California, 2016. <https://doi.org/10.2514/6.2016-5494>.
- [2] Hughes, S., Cheatwood, F., Dillman, R., Calomino, A., Wright, H., DelCorso, J., and Calomino, A., "Hypersonic Inflatable Aerodynamic Decelerator (HIAD) Technology Development Overview," *21st AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar*, Dublin Ireland, 2011. <https://doi.org/10.2514/6.2011-2524>.
- [3] Venkatapathy, E., Hamm, K., Fernandez, I., Arnold, J., Kinney, D., Laub, B., Makino, A., McGuire, M., Peterson, K., Prabhu, D., Empey, D., Dupzyk, I., Huynh, L., Hajela, P., Gage, P., Howard, A., and Andrews, D., "Adaptive Deployable Entry and Placement Technology (ADEPT): A Feasibility Study for Human Missions to Mars," *21st AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar*, Dublin Ireland, 2011. <https://doi.org/10.2514/6.2011-2608>.
- [4] Peterson, K. H., Yount, B., Schneider, N. R., Prabhu, D. K., Arnold, J. O., Squire, T. H., Wercinski, P. F., Chavez-Garcia, J. F., and Venkatapathy, E., "Thermal and Structural Performance of Woven Carbon Cloth For Adaptive Deployable Entry and Placement Technology," *AIAA Aerodynamic Decelerator Systems (ADS) Conference*, Daytona Beach Florida, 2013. <https://doi.org/10.2514/6.2013-1370>.
- [5] NASA, "What's Next?" , Jun. 2017. http://www.nasa.gov/directorates/spacetech/game_changing_development/HIAD.
- [6] NASA, "Adaptable Deployable Entry and Placement Technology (ADEPT)," , Jun. 2017. <https://flightopportunities.nasa.gov/technologies/139>.
- [7] Hollis, B. R., "Surface Heating and Boundary-Layer Transition on a Hypersonic Inflatable Aerodynamic Decelerator," *Journal of Spacecraft and Rockets*, Vol. 55, No. 4, 2018, pp. 856–876. <https://doi.org/10.2514/1.A34046>.
- [8] Brune, A. J., West, T. K., Hosder, S., and Edquist, K. T., "Uncertainty Analysis of Mars Entry Flows over a Hypersonic Inflatable Aerodynamic Decelerator," *Journal of Spacecraft and Rockets*, Vol. 52, No. 3, 2015, pp. 776–788. <https://doi.org/10.2514/1.A33131>.
- [9] Brune, A. J., Hosder, S., and Edquist, K. T., "Uncertainty Analysis of Fluid-Structure Interaction of a Deformable Hypersonic Inflatable Aerodynamic Decelerator," *Journal of Spacecraft and Rockets*, Vol. 53, No. 4, 2016, pp. 654–668. <https://doi.org/10.2514/1.A33532>.
- [10] Brune, A. J., Hosder, S., Edquist, K. T., and Tobin, S. A., "Thermal Protection System Response Uncertainty of a Hypersonic Inflatable Aerodynamic Decelerator," *Journal of Spacecraft and Rockets*, Vol. 54, No. 1, 2016, pp. 141–154. <https://doi.org/10.2514/1.A33732>.
- [11] Kinney, D., "Aerothermal Anchoring of CBAERO Using High Fidelity CFD," *45th AIAA Aerospace Sciences Meeting and Exhibit*, Reno Nevada, 2007. <https://doi.org/10.2514/6.2007-608>.

- [12] Santos, M., Hosder, S., and West, T., “MultiFidelity Modeling for Efficient Aerothermal Prediction of Deployable Entry Vehicles,” *Journal of Spacecraft and Rockets*, Vol. 58, No. 1, 2021, pp. 110–123. <https://doi.org/10.2514/1.A34752>.
- [13] Santos, M., Hosder, S., and West, T., “Multi-Fidelity Turbulent Heating Prediction of Hypersonic Inflatable Aerodynamic Decelerators with Surface Scallop,” *AIAA Aviation 2020 Forum*, AIAA, Virtual, 2020.
- [14] Santos, M., Hosder, S., and West, T., “Multi-Fidelity Heating Prediction of Adaptable, Deployable Entry Placement Technology Vehicles,” *AIAA SciTech 2021 Forum*, AIAA, Virtual, 2021.
- [15] Leifsson, L., Koziel, S., and Tesfahunegn, Y. A., “Multiobjective Aerodynamic Optimization by Variable-Fidelity Models and Response Surface Surrogates,” *AIAA Journal*, Vol. 54, No. 2, 2016, pp. 531–541. <https://doi.org/10.2514/1.J054128>.
- [16] Koziel, S., Tesfahunegn, Y. A., Amrit, A., and Leifsson, L., “Rapid Multi-Objective Aerodynamic Design Using Co-Kriging and Space Mapping,” *57th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, AIAA, San Diego California, 2016.
- [17] Peherstorfer, B., Willcox, K., and Gunzburg, M., “Survey of Multifidelity Methods in Uncertainty Propagation, Inference, and Optimization,” *SIAM Rev.*, Vol. 60, No. 3, 2018, pp. 550–591.
- [18] West, T., and Gumbert, C., “Multifidelity, Multidisciplinary Uncertainty Quantification with Non-Intrusive Polynomial Chaos,” *58th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, AIAA, Grapevine Texas, 2017.
- [19] Qian, E., Peherstorfer, B., O’Malley, D., Vesselinov, V., and Willcox, K., “Multifidelity Monte Carlo Estimation of Variance and Sensitivity Indices,” *SIAM/ASA Journal on Uncertainty Quantification*, Vol. 6, No. 2, 2018, pp. 683–706.
- [20] Sutton, K., and Graves Jr., R. A., “A General Stagnation-Point Convective-Heating Equation for Arbitrary Gas Mixtures,” Tech. Rep. TR R-376, NASA, Nov. 1971.
- [21] Krasnov, N., *Aerodynamics of Bodies of Revolution*, Elsevier, New York, 1970.
- [22] NASA, “NASA Fact Sheet: Space Technology Game Changing Development; Hypersonic Inflatable Aerodynamic Decelerator 2 (HIAD 2) Ground Demonstrations,” , Mar. 2019. https://gameon.nasa.gov/gcd/files/2017/11/HIAD2_FS_170331-5.pdf.
- [23] NASA, “ADEPT for Interplanetary Small Satellite Missions,” , May 2020. <https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20190004912.pdf>.
- [24] Dwyer-Cianciolo, A. M., and Polsgrove, T., “Human Mars Entry, Descent, and Landing Architecture Study: Phase 2 Summary,” *2018 AIAA SPACE and Astronautics Forum and Exposition*, AIAA, Orlando Florida, 2018. <https://doi.org/10.2514/6.2018-5190>.
- [25] Dwyer-Cianciolo, A. M., Polsgrove, T., Sostaric, R. R., Edquist, K. T., Korzun, A. M., and Garcia, J. A., “Human Mars Entry, Descent, and Landing Architecture Study: Phase 3 Summary,” *AIAA Scitech 2020 Forum*, AIAA, Orlando Florida, 2020. <https://doi.org/10.2514/6.2020-1509>.
- [26] Peacocke, L., Bruce, P. J. K., and Santer, M., “Coupled Aerostructural Modeling of Deployable Aerodecelerators for Mars Entry,” *Journal of Spacecraft and Rockets*, Vol. 56, No. 4, 2019, pp. 1221–1230. <https://doi.org/10.2514/1.A34310>.
- [27] Justus, C., James, B., and Johnson, D., “Mars Global Reference Atmospheric Model (Mars-GRAM 3.34): A Programmer’s Guide,” Tech. Rep. TM-108509, NASA, 1996.
- [28] Mazaheri, A., Gnoffo, P., Johnston, C., and Kleb, B., “LAURA User’s Manual,” Tech. Rep. TM 2013-217800, NASA, 2013.
- [29] Smith, A. M., and Cebeci, T., “Numerical Solution of the Turbulent Boundary Layer Equations,” Tech. Rep. TR-DAC-33735, Douglas Aircraft Div., Long Beach California, 1967.
- [30] Johnston, C., and Brandis, A. M., “Modeling of nonequilibrium CO Fourth-Positive and CN Violet emission in CO₂-N₂ gases,” *Journal of Quantitative Spectroscopy and Radiative Transfer*, Vol. 149, No. 1, 2014, pp. 303–317.
- [31] Johnston, C. O., Hollis, B. R., and Sutton, K., “Spectrum Modeling for Air Shock-Layer Radiation at Lunar-Return Conditions,” *Journal of Spacecraft and Rockets*, Vol. 45, No. 5, 2008, pp. 865–878. <https://doi.org/10.2514/1.33004>.
- [32] Johnston, C. O., Hollis, B. R., and Sutton, K., “Non-Boltzmann Modeling for Air Shock-Layer Radiation at Lunar-Return Conditions,” *Journal of Spacecraft and Rockets*, Vol. 45, No. 5, 2008, pp. 879–890. <https://doi.org/10.2514/1.33006>.
- [33] West, T. K., and Brandis, A. M., “Updated Stagnation Point Aeroheating Correlations for Mars Entry,” *2018 Joint Thermophysics and Heat Transfer Conference*, AIAA, Atlanta Georgia, 2018. <https://doi.org/10.2514/6.2018-3767>.

- [34] Forrester, A., Sobester, A., and Keane, A., *Engineering Design via Surrogate Modelling: a Practical Guide*, Wiley, New York, 2008.
- [35] Hicks, R. M., and Henne, P. A., “Wing Design by Numerical Optimization,” *Journal of Aircraft*, Vol. 15, No. 7, 1978, pp. 407–412. <https://doi.org/10.2514/3.58379>.
- [36] West, T. K., and Hosder, S., “Uncertainty Quantification of Hypersonic Reentry Flows with Sparse Sampling and Stochastic Expansions,” *Journal of Spacecraft and Rockets*, Vol. 52, No. 1, 2015, pp. 120–133. <https://doi.org/10.2514/1.A32947>.