

Multi-Fidelity Heating Prediction of Adaptable, Deployable Entry Placement Technology Vehicles

Mario Santos*and Serhat Hosder[†]

Missouri University of Science and Technology, Rolla, MO 65409

Thomas K. West[‡]

NASA Langley Research Center, Hampton, VA, 23681

I. Introduction

FURTHER human and robotic exploration of the solar system presents unique challenges to scientists and engineers in terms of mission and vehicle design. To address some of 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. One of these technologies is the Adaptable, Deployable Entry Placement Technology (ADEPT)[1–3] shown in Fig. 1[4]. As can be seen in Fig. 1, ADEPT vehicles consist of a solid spherical nose cap and a flexible thermal protection system material which remains stowed until atmospheric entry and is deployed mechanically using several ribs, much like an umbrella. 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 vehicle are essential in developing a thermal protection system (TPS) with minimal mass that can survive the EDL mission phase. The TPS also needs to be robust and reliable, which can be achieved through extensive laboratory testing and through uncertainty quantification (UQ) integrated to the analysis and design.

Unlike most legacy entry vehicle designs, ADEPT vehicles are not axisymmetric due to the TPS deployment system. 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 success of the ADEPT development. The completion of the nano-ADEPT SR-1 test flight in 2018 demonstrated the viability of the ADEPT design[5], however much more analysis and investigation is needed to scale up the design to large diameters needed for human exploration.

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.



Figure 1. ADEPT[4] conceptual design.

*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.

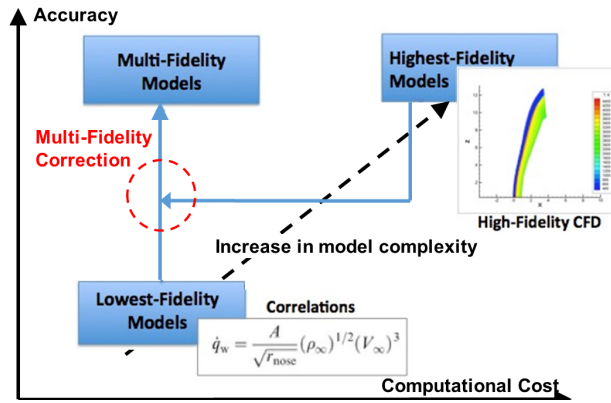


Figure 2. Accuracy vs. computational cost.

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.

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 Hypersonic Inflatable Aerodynamic Decelerator (HIAD) vehicles by Brune et al.[6–8]. 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. A recent study [9] has shown that there is currently no quantitative method to determine whether or not a multi-fidelity approach is appropriate for a given problem a priori. Therefore an investigation of multi-fidelity modeling as a method to reduce the computational cost of the design and analysis of ADEPT vehicles is important. Early work with applying multi-fidelity modeling to hypersonic vehicles was that of aerothermal anchoring utilized by the CBAERO software package[10]. Previous studies by the authors[11, 12] demonstrated that a multi-fidelity approach can produce low cost, accurate predictions of laminar, turbulent, and radiative aerothermal loads on axisymmetric, large diameter deployable entry vehicles such as the HIAD. 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 illustrates that by combining a low cost, less accurate engineering correlation with a more accurate high cost CFD model, multi-fidelity modeling can produce a model 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 [10], wing and airfoil shape optimization[13–15], and uncertainty propagation[15–17]. There are currently several different multi-fidelity methods available that are appropriate for different applications. Adaptive and fusion techniques, such as space-mapping[13] and co-Kriging[14], are primarily used for interpolation and design optimization, while techniques using polynomial chaos expansion[16] and multi-fidelity Monte Carlo estimation[17] are used primarily for uncertainty quantification and propagation. This study will use the co-Kriging based multi-fidelity modeling approach previously developed for large diameter re-entry vehicles in both laminar and turbulent flow[11, 12].

The objective of this work is to implement and demonstrate a co-Kriging based multi-fidelity (CKMF) modeling methodology for the prediction of laminar and turbulent convective heating on large diameter ADEPT vehicles in Mars entry. The multi-fidelity surrogate models for laminar and turbulent convective heat flux will be constructed as a function of velocity, density, and nose radii to predict aerothermal loads along various entry trajectories and ADEPT geometries. This research will utilize several computational tools of various fidelity levels such as the Sutton-Graves [18] with Krasnov [19] 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. Due to the vehicle axial asymmetry, CKMF models will be built for the heat flux distributions at several circumferential locations on the vehicle.

The following section describes the ADEPT geometries used and the determination of the sample space with a six degree-of-freedom Mars entry model. Section III describes the computational tools used to construct the multi-fidelity surrogate model. Section IV describes the co-Kriging approach and methodology used in this research. Section V contains the current CKMF model results for the laminar and turbulent convective heat flux and the final section includes key conclusions and future work.

II. Vehicle Geometry and Sample Space

The first step of this study was to define and determine the ADEPT vehicle geometry parameters. The ADEPT vehicle concept consists of a solid nose cap forward of the payload section, surrounded by a flexible TPS that is deployed mechanically using several ribs and is attached to the solid nose cap. An illustration of the ADEPT can be seen in Fig. 3. The ADEPT geometries used for this study are based on the 18.8 m large diameter Mars ADEPT geometries described in NASA's Exploration Feed-Forward[20] (EFF) study. The geometric parameters describing the vehicle shape are shown in Fig. 4. For this study, the ADEPT vehicles used had a base radius (R_b) of 9.4 m, shoulder radius (R_s) of 0.47 m, and half-cone and shoulder turn angles (δ and θ respectively) both 70° . A nose radius (R_n) range of 4 to 20 m was chosen as the nose radius space.



Figure 3. ADEPT vehicle diagram[21].

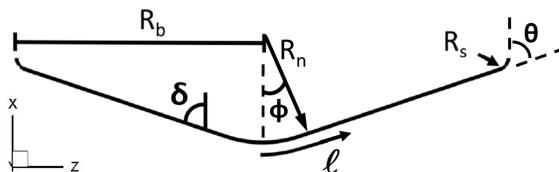


Figure 4. ADEPT geometry parameters.

For this study, the ADEPT vehicles have twelve ribs used to deploy the flexible TPS with each rib having a width of 20 cm and 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.[22], and an example ADEPT geometry used in this study is illustrated in Fig. 5. 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 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.

To calculate the entry trajectories, the ADEPT vehicles were assumed to be in ballistic entry with 0° angle of attack. Again using the EFF study as a guide for the mass and entry velocity ranges, the six degree-of freedom (DOF) model utilized by Brune et al.[6] was used to determine the trajectory range for a ADEPT vehicle in Martian entry. In this model, the spacecraft was assumed to enter the atmosphere at 0 degrees longitude, 0 degrees latitude with an easterly heading of 90 degrees. The flight path of the vehicle was assumed to be at an angle of -15 degrees with an entry interface altitude of 104 km above the surface. The minimum and maximum entry velocities used in the trajectory calculations were 3.35 to 7.3 km/s, respectively, and the minimum and maximum vehicle masses used were 7.2 to 40 MT, respectively. The resulting bounding entry trajectories are shown in Fig. 6. For each trajectory point, the Sutton-Graves correlation [18], shown in Eq. 1, was used to calculate the cold-wall stagnation point heat flux ($\dot{q}$). Eq. 1 assumes a gas mixture of 97% CO_2 and 3% N_2 by mass in chemical

equilibrium flowing over an axisymmetric blunt body in terms of the density (ρ) and velocity (V):

$$\dot{q} = 1.83 \times 10^{-4} \sqrt{\rho/R_n} V^3 \quad [W/m^2] \quad (1)$$

The velocity-density space chosen for this study is illustrated by the black dashed line in Fig. 6. This space was chosen because it contains all of the predicted maximum heating locations while being low enough in altitude to be in continuum flow and high enough in altitude that an aerodynamic decelerator would still be feasible to be in use. The chosen three dimensional density, velocity, and nose radius space used for this study is summarized in Table 1.

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	6.6752×10^{-5}
Maximum	20	7	1×10^{-3}

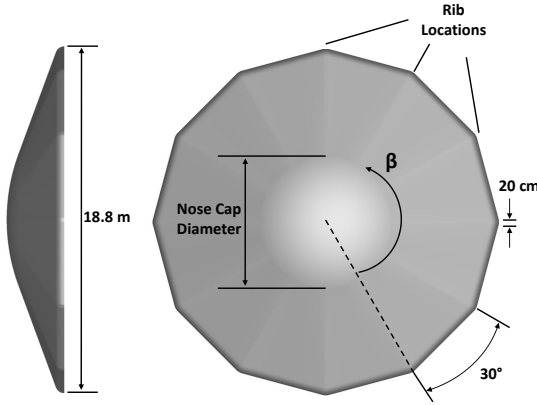


Figure 5. Example ADEPT vehicle used in this study.

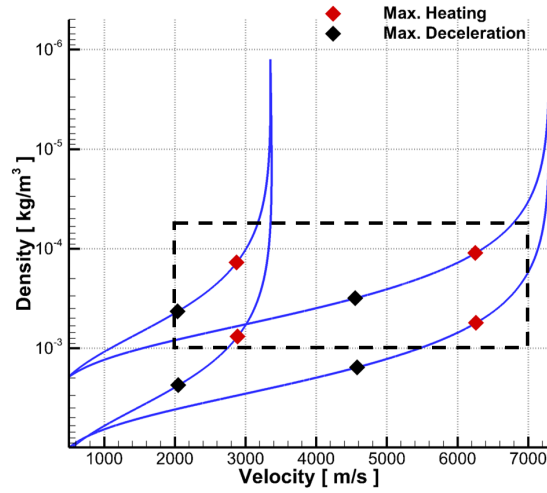


Figure 6. Velocity-density trajectory profiles.

III. Computational Models

III.A. LAURA CFD Predictions

NASA Langley Research Center's high-speed thermochemical non-equilibrium CFD software LAURA is used as the high-fidelity computational model for this study to obtain both fully laminar and fully turbulent boundary layer 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 [23] and to cluster the grid cells near the wall to obtain wall cell Reynolds numbers of approximately one. The flow field was assumed to be steady state with a two temperature, thermochemical nonequilibrium gas model. The turbulence model used was the Cebeci-Smith algebraic model[24]. The Martian atmosphere was assumed to be composed of 97.417% CO₂ and 2.583% N₂ by mass with the flow field modeled by a 10 species composition model: CO₂, CO, N₂, O₂, NO, C, N, O, CN, and C₂. The reaction finite rate chemistry model that was used is given by Johnston and Brandis [25]. This model was tuned to match CO Fourth-Positive and CN Violet emission measured in the EAST shock-tube experiments.

The grids used for this study are half-geometry structured grids with an O-H grid topology for the vehicle surface to allow for accurate flow modeling at the stagnation point. Each grid has approximately 1.84×10^6 cells determined using a grid convergence study given in the Results Section V.A. The grid is divided into two blocks: a rectangular grid block at the stagnation region, and a polar grid block outside of the stagnation region. Both grid blocks have 96 cells normal to the vehicle surface with the rectangular grid region having dimensions of $48 \times 96 \times 96$ cells, and the polar grid region having dimensions of $76 \times 192 \times 96$ cells. Adaptation

of the grid near the wall to produce cell wall Reynolds numbers near one ensures that the first cell height off the wall is at $y^+ \approx 1$. A cold wall boundary condition was used for the vehicle surface along with a super-catalytic surface chemistry model. 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 surface grid used in this study and corresponding laminar heat flux distribution are shown in Fig. 7. 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. Using 108 2.2 GHz Intel Xeon processors, a fully converged high-fidelity solution requires approximately 60 hours of computation time.

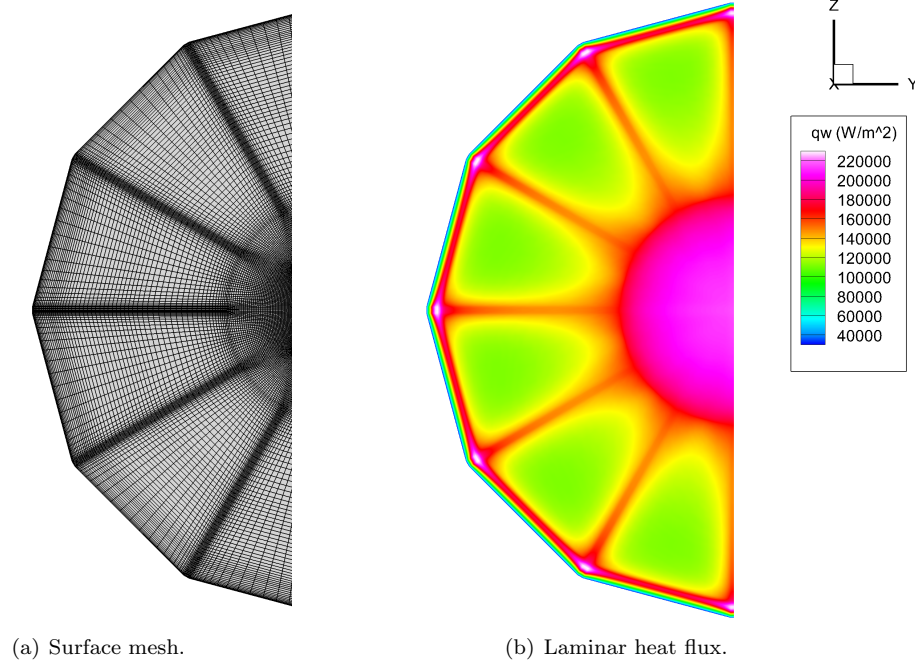


Figure 7. Example ADEPT surface mesh and corresponding laminar heat flux.

III.B. Sutton-Graves with 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 centerline 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. 1. For heating downstream of the stagnation point, correlations for the ratio of heating at an arbitrary point to the stagnation point from Krasnov[19] are used. The body is separated into its sphere and cone regions, using different correlations for the heating ratio, $\frac{\dot{q}}{\dot{q}_0}$, for each region. For the spherical region, the following is used in terms of freestream Mach number, M_∞ ,

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

where,

$$\zeta = 1 - \frac{1}{\gamma M_\infty^2}, \quad (3)$$

$$f_1(\phi) = \phi^2 - \phi \sin 2\phi + \sin^2 \phi, \quad (4)$$

$$f_2(\phi) = \frac{1}{2} \left[\frac{3}{2} \phi^2 + \phi \sin 2\phi \left(\frac{3}{2} + \sin^2 \phi \right) + \frac{3}{2} \sin^2 \phi + \frac{1}{2} \sin^4 \phi \right]. \quad (5)$$

Here, γ is the frozen freestream specific heat ratio and ϕ is the surface inclination angle measured from the normal direction at the nose to a point on the spherical region, as shown in Fig. 4. Next, the heating ratio on the conical surface is modeled by,

$$\frac{\dot{q}}{\dot{q}_0} = \frac{A\bar{\ell}}{\sqrt{B + \bar{\ell}^3}}. \quad (6)$$

where,

$$A(\delta) = \frac{\sqrt{3}}{2} \left\{ \left[\left(1 - \frac{1}{\gamma M_\infty^2} \right) \sin^2 \delta + \frac{1}{\gamma M_\infty^2} \right] \left(\frac{\pi}{2} - \delta \right) \right\}^{\frac{1}{2}}, \quad (7)$$

$$B(\delta) = \frac{\frac{3}{16} D(\phi) / \phi}{\left(1 - \frac{1}{\gamma M_\infty^2} \right) \sin^4 \delta + \frac{\sin^2 \delta}{\gamma M_\infty^2}} - \cot^3 \delta, \quad (8)$$

$$D(\phi) = \left(1 - \frac{1}{\gamma} \right) \left(\phi^2 - \frac{\phi}{2} \sin(4\phi) + \frac{1}{8} (1 - \cos(4\phi)) \right) + \frac{4}{\gamma} \left(\phi^2 - \phi \sin(2\phi) + \frac{1}{2} (1 - \cos(2\phi)) \right) \quad (9)$$

$$\bar{\ell} = \frac{\ell + s}{R_n}. \quad (10)$$

Here, ℓ is the running length as shown in Fig. 4. To ensure a continuous heat flux distribution, an offset to the running length 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 s using the secant method. Then, with the offset found, the final heat flux over the body can be determined. Utilizing a single 2.2 GHz Intel Xeon processor, a single Sutton-Graves-Krasnov solution for convective heating requires approximately 2 seconds of computation time.

IV. Multi-Fidelity Model

Co-Kriging is an interpolative method that has also been used extensively in low-speed aerospace applications [13, 15]. For this study, a co-Kriging based multi-fidelity surrogate modeling approach was used that was developed previously by Santos et al.[11, 12] for the prediction of the laminar and turbulent aerothermal response of deployable entry vehicles. In this approach, several refinements to co-Kriging approach were implemented to increase computational speed and accuracy. These refinements include adaptive sampling, LU decomposition, and distribution shape parameterization.

IV.A. Co-Kriging Modeling

Co-Kriging modeling is a direct extension of Kriging modeling and therefore a brief overview of the Kriging method is necessary for the discussion of co-Kriging. The co-Kriging formulation used for this study is the same auto-regressive form given by Forrester et al.[26].

Kriging models are surrogate models used to interpolate an unknown function, f , given n number of data points. In Kriging models the interpolated data points are modeled by a Gaussian process in the form of $\mathbf{y} = \mu + \mathbf{b}\Psi$, where y is a column vector of model responses, μ is the maximum likelihood estimate (MLE), b is a row vector of coefficients, and Ψ is a correlation matrix between the sampled data points, $x^{(i)}$, and the interpolated data points, x^{n+1} , with k dimensions. In the Kriging approach, each element of Ψ is a radial basis function of a specific form given in Eq. 11.

$$\psi^{(i)} = \exp \left(- \sum_{j=1}^k \lambda_j |x_j^{n+1} - x_j^{(i)}|^{p_j} \right) \quad (11)$$

Where λ and p are coefficients to be determined. Thus, the covariance matrix is given by Eq. 12.

$$\text{Cov}(\mathbf{y}, \mathbf{y}) = \sigma^2 \Psi \quad (12)$$

where

$$\sigma^2 = \frac{(\mathbf{y} - \mathbf{1}\mu)^T \Psi^{-1} (\mathbf{y} - \mathbf{1}\mu)}{n} \quad (13)$$

and

$$\mu = \frac{\mathbf{1}^T \Psi^{-1} \mathbf{y}}{\mathbf{1}^T \Psi^{-1} \mathbf{1}} \quad (14)$$

An estimate of the accuracy of the surrogate can be determined from the log-likelihood given by Eq. 15.

$$\ln(L) = -\frac{n}{2} \ln(\sigma^2) - \frac{1}{2} \ln|\Psi| \quad (15)$$

To create a Kriging model, the parameters λ , and p in Eq. 11 are solved for by finding the λ and p that maximize the log of the likelihood, L . Once the parameters λ and p are known, the response at any point can be interpolated. The process of creating a co-Kriging model is very similar to the process of creating a Kriging model. In the co-Kriging model, the co-variance matrix is given by Eq. 16. Here, two sets of parameters (λ_{LF} , P_{LF} , ρ , λ_d and P_d) corresponding to Ψ_{LF} and Ψ_d will need to be determined to create the co-Kriging model.

$$\mathbf{C} = \begin{pmatrix} \sigma_{LF}^2 \Psi_{LF}(\mathbf{X}_{LF}, \mathbf{X}_{LF}) & \rho \sigma_{LF}^2 \Psi_{LF}(\mathbf{X}_{LF}, \mathbf{X}_{HF}) \\ \rho \sigma_{LF}^2 \Psi_{LF}(\mathbf{X}_{HF}, \mathbf{X}_{LF}) & \rho^2 \sigma_{LF}^2 \Psi_{LF}(\mathbf{X}_{HF}, \mathbf{X}_{HF}) + \sigma_d^2 \Psi_d(\mathbf{X}_{HF}, \mathbf{X}_{HF}) \end{pmatrix} \quad (16)$$

First, a Kriging model of the low-fidelity data is first built to obtain λ_{LF} and P_{LF} . Then, a difference model, $d = y_{HF} - \rho y_{LF}(x_{HF})$, is constructed where $y_{LF}(x_{HF})$ are the values of y_{LF} at locations common to those of x_{HF} . Applying the Kriging method to this difference model results in a log of the likelihood of the difference model, L_d , given by:

$$\ln(L_d) = -\frac{n_{HF}}{2} \ln(\sigma_d^2) - \frac{1}{2} \ln|\Psi_d(\mathbf{X}_{HF}, \mathbf{X}_{HF})| \quad (17)$$

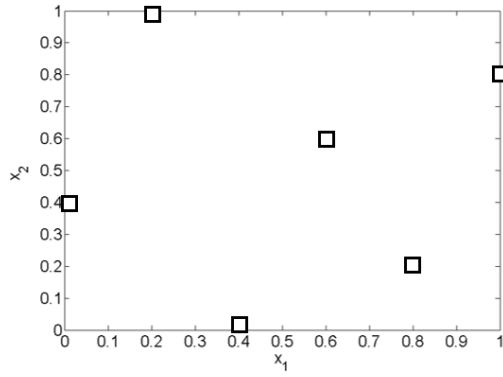
Next, ρ , λ_d , and P_d are then found by maximizing Eq. 17. Once the optimum ρ , λ_{LF} , P_{LF} , λ_d , and P_d are found the co-Kriging model can be used to interpolate the response at any point.

IV.B. Sampling Method

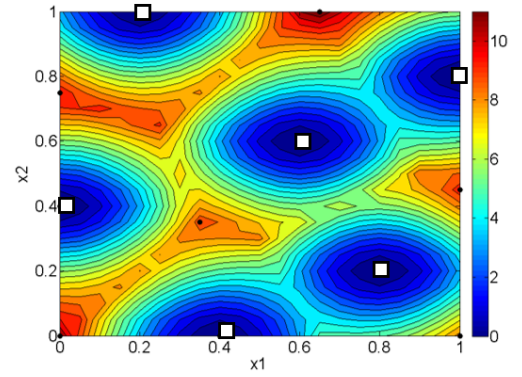
In this study, the low-fidelity model was sampled linearly in each dimension. However, co-Kriging models are very sensitive to the number and location of the high-fidelity data points used to train the model. Choosing appropriate sample locations can minimize the number of high-fidelity samples needed to achieve a global level of accuracy, which reduces overall computational cost. Thus, the sampling plan used to build the model is a very important part of the co-Kriging process. The sampling method used in this study is a distance weighted root mean square error (RMSE)[26] adaptive sampling scheme. Along with the function response, the co-Kriging model can also estimate the RMSE of the prediction at any point. In this sampling scheme, a minimum number of data points, N_{min} , are chosen and a space filling Latin hypercube is found using N_{min} . Next, the high-fidelity model at the training points are evaluated, and the Co-Kriging model is built using these data points. This co-Kriging model is then used to predict both the high-fidelity model response and the RMSE over the 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. An example of this sampling scheme is shown in Fig. 8.

IV.C. Surface Distribution Parameterization

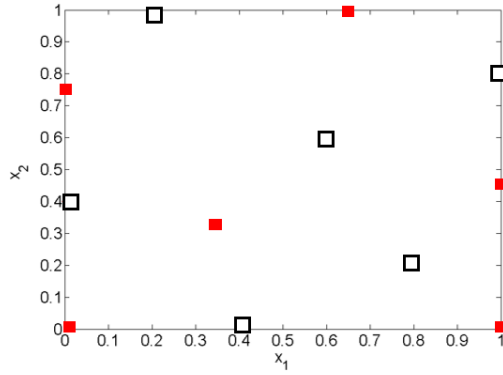
A study by Santos et al.[11] demonstrated that applying the CKMF modeling technique to Hicks-Henne parameterized[27] output distributions instead of each surface point individually resulted in predictions that were more computationally efficient and accurate. This study also uses the Hicks-Henne parameterization technique to parameterize the output surface convective heat flux distributions. The Hicks-Henne shape parameterization technique is commonly used in aerospace engineering applications for parameterization of



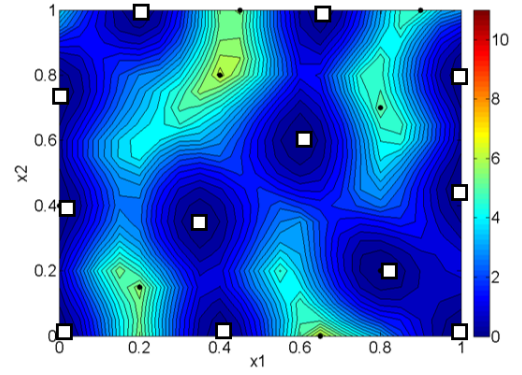
(a) Generate initial sample locations (white squares).



(b) Build co-Kriging model and generate distance weighted RMSE data.



(c) Add locations of distance weighted RMSE maxima (red squares) to sample set.



(d) Build new co-Kriging model and generate new distance weighted RMSE data.

Figure 8. Example of distance weighted RMSE sampling technique.

airfoil shapes [27]. In this technique, a set of n control points on a baseline shape are chosen. A linear combination of sine functions over the n control points, shown in Eq. 18, are used to perturb the baseline shape, $y(x)^i$, to get the desired “deformed” shape, $y(x)$. Here, a_j are amplitude coefficients, b_j are width coefficients, and c_j are the x -locations of the control points. In this study, the baseline shape was assumed to be a straight line at a constant value of one.

$$y(x) = y^i(x) + \sum_{j=0}^n a_j \sin^{b_j}(\pi x^{\ln(0.5/c_j)}) \quad (18)$$

Hicks-Henne bump functions work well when the original shape is subject to small deformations of less than one. Thus, for best results, the surface distributions should be scaled to be approximately between zero and one. For convective heating, this scaling was achieved by scaling the distributions by the stagnation value. The 22 control point locations used in the previous study by Santos et al.[11] were also used in this study. An example of the Hicks-Henne bump function being used to approximate a convective heating distribution can be seen in Fig. 9. Applying the CKMF method to the Hicks-Henne shape parameterization technique requires only 23 separate CKMF models for the convective heat flux, one model for the stagnation value, and 22 models for the control points.

IV.D. Application of CKMF Process to ADEPT

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 actually be modeled. Thus, to apply the described CKMF approach to the ADEPT, each geometry is evaluated along four lines across the surface: 0deg, 5deg, 10deg, and 15deg, 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, resulting in four different CKMF models. Interpolation is then used to obtain the heat flux between slices.

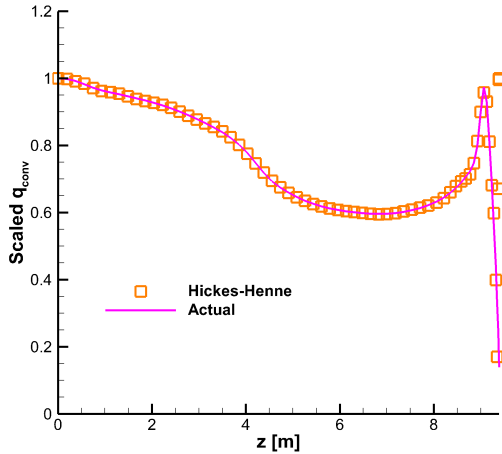


Figure 9. Hicks-Henne approximation of a scaled surface convective heat flux distribution at surface slice location.

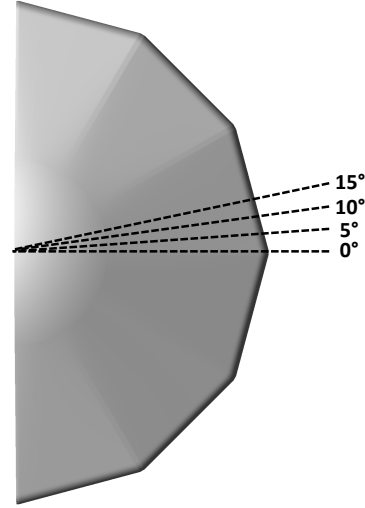


Figure 10. Solution slice locations for CKMF process.

V. Preliminary Results and Discussion

First, the appropriate grid resolution for the high-fidelity LAURA CFD model had to be determined by performing a grid convergence study. Next, the performance of the CKMF approach in predicting the convective heating on ADEPT vehicles in Martian entry needs to be evaluated. This work is currently in progress and the preliminary results are included in this abstract. To evaluate the model performance, the CKMF model will be evaluated at select test points and compared to the high-fidelity LAURA CFD results at these points. The CKMF approach will be used to model the laminar and turbulent convective heat flux distributions on ADEPT vehicles across the chosen sample space detailed in section II.

V.A. CFD Grid Convergence Study

To determine an appropriate grid resolution for the high-fidelity CFD model that resulted in accurate convective heat flux predictions while minimizing computational cost, a grid convergence study was performed. For this grid convergence study two different cases were used: a low heat flux case and a high heat flux case. The high heat flux case used was a 4 m nose radius geometry at a velocity of 6.0 km/s and atmospheric density of 1.9710×10^{-4} kg/m³. The low heat flux case used was a 20 m nose radius geometry at a velocity of 3.0 km/s and atmospheric density of 3.3868×10^{-4} kg/m³. The quantities of interest analyzed in this grid convergence study were stagnation point heat flux and the total surface heating (defined as the integral of the heat flux over the entire surface) given in Eq. 19.

$$Q = \int_S \dot{q}_{conv} dA \quad [W] \quad (19)$$

Five grid sizes were used in this convergence study. These grid sizes are coarse, medium-coarse, medium, medium-fine, and fine with total cell sizes of 1.5360×10^5 , 2.3040×10^5 , 1.2288×10^6 , 1.8432×10^6 , and 9.8304×10^6 cells, respectively. The results of the convergence study are given in Table 2 and are plotted in Fig. 11. In both Table 2 and Fig. 11, the quantities of interest are normalized by the medium grid value for ease of interpretation. These results show that there is very little change in accuracy from the medium-fine to the

fine grid levels while the computational cost increases by a factor of four. Thus, the medium-fine grid level was chosen as the grid size to be used in this study.

Table 2. Grid convergence study results.

Low Heating Case					
Quantity of Interest	Coarse	Medium-Coarse	Medium	Medium-Fine	Fine
Stagnation Heat Flux	1.0616	1.0308	1.0	0.99176	0.98670
Total Heating	1.0838	1.0464	1.0	0.99200	0.98020
High Heating Case					
Quantity of Interest	Coarse	Medium-Coarse	Medium	Medium-Fine	Fine
Stagnation Heat Flux	1.2364	1.0442	1.0	0.98582	0.97930
Total Heating	1.1390	1.0570	1.0	0.97504	0.96418

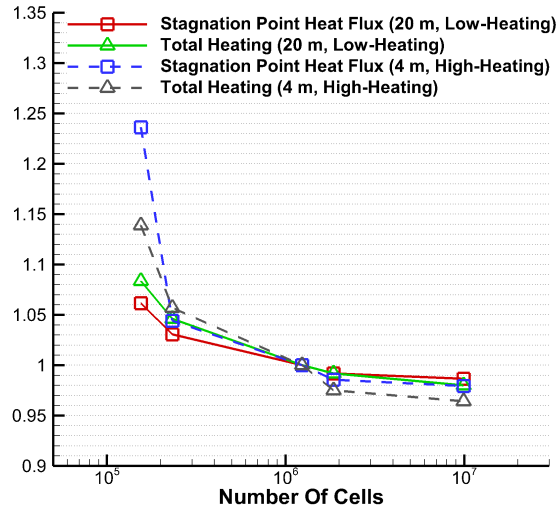


Figure 11. Grid convergence study results.

V.B. ADEPT Vehicle in Mars Entry

To evaluate the performance of the ADEPT CKMF surrogates, 36 random test cases across the design space have been chosen using a Latin hypercube sampling scheme. These test case locations are given in Table 3. The freestream Reynolds number ranged between 0.2308×10^6 and 5.969×10^6 [1/m] and the Mach number ranged between 9.59 and 33.06 for these test cases. Note that the test cases will be used exclusively to assess the performance of the CKMF models and will not be used in the construction of the CKMF models. For all CKMF models in this study, 512 low-fidelity training points (8 points in each design space dimension) will be used to construct the CKMF models. This number of low-fidelity training points was determined to be optimal in a previous study by the authors[11]. Note that the low-fidelity model is the same for both the laminar and turbulent heating predictions.

In the final manuscript, the percent error in the CKMF model prediction of convective heat rate per unit depth for each test case will be calculated using Eq. 20. In this equation, $\dot{q}_{MF}$ is the multi-fidelity surface heat flux distribution, $\dot{q}_{HF}$ is the high-fidelity surface heat flux distribution, and $\dot{q}_{0,HF}$ is the high-fidelity stagnation point heat flux. The average percent error across all 36 test cases will be used as a performance metric in a CKMF model convergence study as the number of high-fidelity training cases is increased.

$$\% \text{ Error} = \frac{1}{R_b} \int \left| \frac{\dot{q}_{MF} - \dot{q}_{HF}}{\dot{q}_{0,HF}} \right| dz \times 100 \quad (20)$$

Table 3. Test cases used in model performance evaluation.

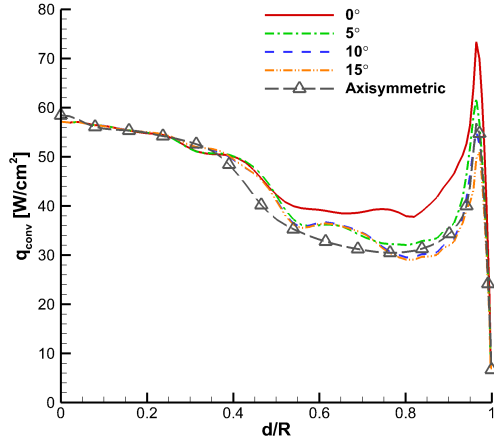
Test Case	R_n [m]	V [km/s]	ρ [kg/m ³]	Test Case	R_n [m]	V [km/s]	ρ [kg/m ³]
1	11.77	3.000	9.2558x10 ⁻⁴	19	18.17	6.428	4.2711x10 ⁻⁴
2	20.00	4.428	3.3867x10 ⁻⁴	20	9.02	6.285	3.9532x10 ⁻⁴
3	9.48	5.714	6.6752x10 ⁻⁵	21	11.31	4.285	6.2874x10 ⁻⁴
4	8.11	3.714	1.2392x10 ⁻⁴	22	10.40	2.857	3.6590x10 ⁻⁴
5	16.80	4.000	9.0952x10 ⁻⁵	23	10.85	4.857	1.0616x10 ⁻⁴
6	7.20	3.142	7.3391x10 ⁻⁴	24	5.37	3.285	1.6885x10 ⁻⁴
7	5.82	5.000	5.3865x10 ⁻⁴	25	8.57	2.000	9.8265x10 ⁻⁵
8	16.34	6.000	1.1470x10 ⁻⁴	26	6.74	5.857	1.0000x10 ⁻³
9	4.45	6.142	7.7918x10 ⁻⁵	27	19.54	6.571	1.3388x10 ⁻⁴
10	12.68	6.857	8.4182x10 ⁻⁵	28	7.65	5.428	2.4856x10 ⁻⁴
11	4.00	4.142	6.7930x10 ⁻⁴	29	17.25	3.857	7.9294x10 ⁻⁴
12	6.28	2.142	2.9014x10 ⁻⁴	30	13.60	6.714	2.3006x10 ⁻⁴
13	18.62	5.142	7.2119x10 ⁻⁵	31	19.08	3.571	1.9709x10 ⁻⁴
14	14.97	2.714	1.4465x10 ⁻⁴	32	13.14	2.428	4.6146x10 ⁻⁴
15	9.94	2.571	1.8242x10 ⁻⁴	33	15.88	3.428	4.9856x10 ⁻⁴
16	15.42	5.571	8.5670x10 ⁻⁴	34	17.71	2.285	2.6855x10 ⁻⁴
17	12.22	7.000	5.8195x10 ⁻⁴	35	4.91	4.571	2.1294x10 ⁻⁴
18	14.51	4.714	3.1347x10 ⁻⁴	36	14.05	5.285	1.5628x10 ⁻⁴

V.B.1. Laminar Heating Prediction

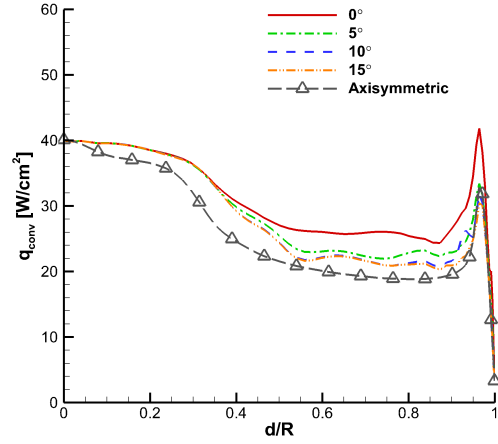
To date, 18 high-fidelity laminar LAURA CFD training cases have been completed. These training cases were chosen using the distance weighted RMSE sampling plan outlined previously. Heat flux distribution results for the laminar heating CKMF models at selected test conditions using 18 high-fidelity training points are shown in Fig. 12. Also included in this figure are high-fidelity CFD results for an axisymmetric sphere cone geometry for comparison. Note that currently none of the high-fidelity CFD ADEPT test cases have been evaluated and are not included in Fig. 12. The final manuscript will include the high-fidelity ADEPT results for the selected test cases. Comparing the CKMF prediction to the high-fidelity axisymmetric results shows that the CKMF model does well in predicting the stagnation point heat flux using 18 high-fidelity training cases. However, the heat flux distributions have not converged and additional high-fidelity training cases are being evaluated for the construction of the CKMF model until convergence is achieved. Previous studies by the authors have shown that for axisymmetric vehicles, the CKMF models require between 24 and 36 high-fidelity training cases to converge[11, 12].

VI. Conclusion and Work to be Done

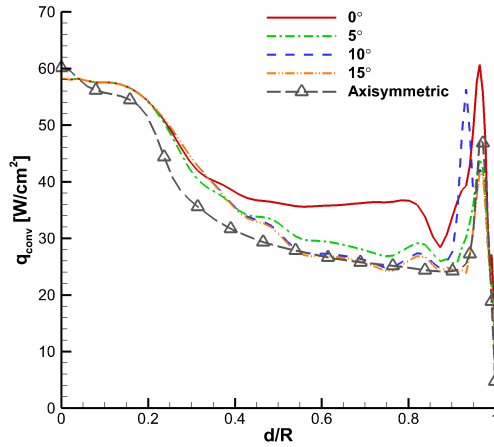
The objective of this work is to implement and demonstrate a co-Kriging based multi-fidelity (CKMF) modeling methodology for the prediction of laminar and turbulent convective heating on ADEPT geometries during Mars entry. Preliminary results for laminar heating show that the CKMF approach predicts the stagnation point heat flux well using 18 high-fidelity training cases, however more high-fidelity training cases are required to converge the CKMF model of heat flux over the entire surface. The final manuscript will have convergence studies of the laminar and turbulent heating CKMF models versus the number of high-fidelity training cases. The high-fidelity CFD ADEPT results for the test cases will also be included in the final manuscript. Finally, the finished manuscript will have an analysis of the computational cost of the CKMF models compared to the low and high-fidelity models.



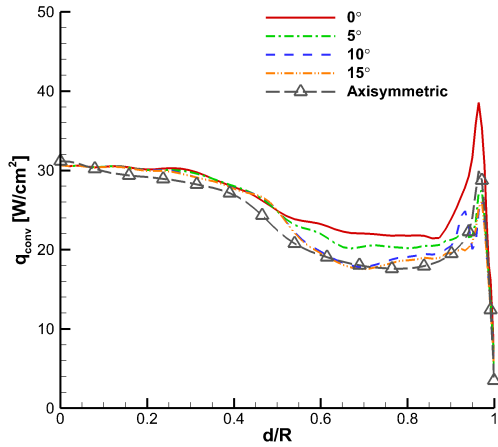
(a) CKMF laminar convective heating prediction for test case condition 17.



(b) CKMF laminar convective heating prediction for test case condition 20.



(c) CKMF laminar convective heating prediction for test case condition 26.



(d) CKMF laminar convective heating prediction for test case condition 30.

Figure 12. CKMF Laminar convective heating predictions for selected test case conditions.

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).

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] 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>.
- [3] 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>.
- [4] NASA, “Adaptable Deployable Entry and Placement Technology (ADEPT),” , Jun. 2017. <https://flightopportunities.nasa.gov/technologies/139>.
- [5] Cassell, A., Wercinski, P., Smith, B., Yount, B., Nishioka, O., and Kruger, C., “ADEPT Sounding Rocket One Flight Test Overview,” *AIAA Aviation 2019 Forum*, Dallas Texas, 2019. <https://doi.org/10.2514/6.2019-2896>.
- [6] 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>.
- [7] 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>.
- [8] 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, 2017, pp. 141–154. <https://doi.org/10.2514/1.A33732>.
- [9] Giselle Fernández-Godino, M., Park, C., Kim, N. H., and Haftka, R. T., “Issues in Deciding Whether to Use Multifidelity Surrogates,” *AIAA Journal*, Vol. 57, No. 5, 2019, pp. 2039–2054. <https://doi.org/10.2514/1.J057750>.
- [10] 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>.
- [11] Santos, M., Hinkle, A., Hosder, S., and West, T., “Multi-Fidelity Modeling for Efficient Aerothermal Prediction of Deployable Re-Entry Vehicles,” *22nd AIAA International Space Planes and Hypersonics Systems and Technologies Conference*, AIAA, Orlando Florida, 2018. <https://doi.org/10.2514/6.2018-5268>.
- [12] Santos, M., Hosder, S., and West, T., “Multi-Fidelity Turbulent Heating Prediction of Hypersonic Inflatable Aerodynamic Decelerators with Surface Scalloping,” *AIAA Aviation 2020 Forum*, AIAA, Reno Nevada, 2020.
- [13] 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>.
- [14] 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.
- [15] Peherstorfer, B., Willcox, K., and Gunzburger, M., “Survey of Multifidelity Methods in Uncertainty Propagation, Inference, and Optimization,” *SIAM Rev.*, Vol. 60, No. 3, 2018, pp. 550–591.
- [16] 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.
- [17] 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.

- [18] 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.
- [19] Krasnov, N., *Aerodynamics of Bodies of Revolution*, Elsevier, New York, 1970.
- [20] Dwyer-Ciancolo, A., Davis, J., Engelund, W., Komar, D., Queen, E., Samareh, J., Way, D., and Zang, T., “Entry, Descent, and Landing Systems Analysis Study: Phase 2 Report on Exploration Feed-Forward Systems,” Tech. Rep. TM-217055, NASA, Feb. 2011.
- [21] NASA, “ADEPT for Interplanetary Small Satellite Missions,” , May 2020. <https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20190004912.pdf>.
- [22] 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>.
- [23] Mazaheri, A., Gnoffo, P., Johnston, C., and Kleb, B., “LAURA User’s Manual,” Tech. Rep. TM 2013-217800, NASA, 2013.
- [24] 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.
- [25] 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.
- [26] Forrester, A., Sobester, A., and Keane, A., *Engineering Design via Surrogate Modelling: a Practical Guide*, Wiley, New York, 2008.
- [27] 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>.