Coupling a Computational Fluid Dynamics Model to a Spacecraft Thermal System Model for the DraMS Instrument Thermal Analysis

Daniel G. Bae[1] and David E. Steinfeld[2]
NASA Goddard Space Flight Center, Greenbelt, MD 20771

The Dragonfly Mass Spectrometer (DraMS) is an instrument on the Dragonfly mission, which will spend 7 years in deep space cruise before landing and operating on the surface of Titan. Vacuum thermal analyses are required for deep space cruise, and convection analyses are required for the Titan surface operations. Model exchanges across multiple thermal teams are needed for all phases of the mission. For DraMS, Thermal Desktop® (TD) has been the main thermal analytical tool of choice due to its capability in modeling complex thermal systems with relatively low computational power and for its availability across thermal teams. However, TD does not have computational fluid dynamics (CFD) capability and struggles to accurately capture complex convective behavior. DraMS has fans operating in tandem and gas flow behaviors are not easily predicted due to its complex flow paths. CFD software, such as Fluent, can model and predict such complex flow behaviors, but CFD models are computationally expensive, and its workflow processes are not tailored towards simulating large and complex systems. Therefore, a coupled modeling approach was chosen for DraMS: A TD model was used for simulating all the conductive, radiative, and source terms, while a Fluent CFD model was added on, as needed, to the TD model to provide the convective boundary conditions using the System Coupling software. The coupling software allows the TD and Fluent models to communicate data and arrive at a co-solved and co-converged solution. Furthermore, Thermal Iso-value Exchange (TIE) method was developed to facilitate and improve the TD-Fluent data exchange process. This paper will discuss the analytical studies that were done to verify the accuracy and usability of the coupled approach and the challenges associated, which lead to the development of the TIE approach. DraMS thermal design and co-solved analysis results will also be discussed.

Nomenclature

g	=	gravitational acceleration, m/s^2
h	=	heat transfer coefficient, W/m^2·K
$\dot{Q}_{conv}$	=	surface convective heat flow, W
$\dot{Q}_{Fluent}$	=	total surface convective heat flow calculated from Fluent, W
$\dot{Q}_{TD}$	=	total surface convective heat flow calculated from TD, W
Pr	=	Prandtl number, dimensionless
T_{ref}	=	local convective air temperature, K
T_{surf}	=	temperature of the convecting surface, K

Acronyms and Abbreviations

API	=	application programming interface
Atm	=	unit of pressure equivalent to 1 Earth atmosphere of pressure at sea level (101,325 Pa)
CFD	=	computational fluid dynamics
CFL	=	Courant-Friedrichs-Lewy
DraMS	=	Dragonfly mass spectrometer

[1] DraMS Lead Thermal Analyst, Thermal Engineering Branch 545
[2] DraMS Thermal Product Development Lead, Thermal Engineering Branch 545

GC	=	gas chromatograph
GCMS	=	gas chromatograph mass spectrometry
GPS	=	gas processing system
HT-T	=	convective coefficient (h) with local air temperature (T_{ref}) and surface temperature transfer method
ITMS	=	ion trap mass spectrometer
LDMS	=	laser desorption mass spectrometry
MEB	=	main electronics box
MSP	=	miniature scroll pump
N-S	=	Navier-Stokes
Q-N	=	quasi-Newton stability method
Q-T	=	convective heat flux and surface temperature transfer method
RF	=	radio frequency
SCTAS	=	spacecraft thermal analysis software
STOP	=	structural thermal optical performance
SyC	=	system coupling
TD	=	Thermal Desktop®
TIE	=	thermal iso-value exchange

I. Introduction and Motivation

THE Dragonfly Mass Spectrometer (DraMS) will be operating on the surface of Titan, Saturn's largest moon, to study its organic-rich ocean world[1]. Titan's cold (94 K/-179 °C) and dense atmosphere (1.5 atm), and low gravity ($g=1.35\ m/s^2$) adds a challenging facet to the already difficult thermal engineering of the DraMS instrument. The DraMS instrument requires heavy focus on all three modes of heat transfer, conduction, convection, and radiation. With the many hot pipes and valves ($>100°C$) required for its gas processing system (GPS), and the cold components such as the laser ($25\pm1°C$) for the Laser Desorption Mass Spectrometry (LDMS) and the sample cups (<170 K or -$163°C$) sharing the same instrument volume, conductive isolation is important for the proper functionality of the instrument. With the pyrolysis oven used for the Gas Chromatograph Mass Spectrometry (GCMS) and for the thermal analysis of the instrument during its long deep space cruise of 7 years, thermal radiation analysis cannot be overlooked. With the dense atmosphere of Titan and high-power density of the DraMS instrument (>300 W), detailed convective analysis is crucial and has been highlighted in Bae et al[2]. The overview of the convective cooling scheme of the DraMS instrument is shown in Figure 1, which shows how the Titan gas is flowed through the instrument via fans to keep the instrument cool during its science operations, such as the GCMS and LDMS modes.

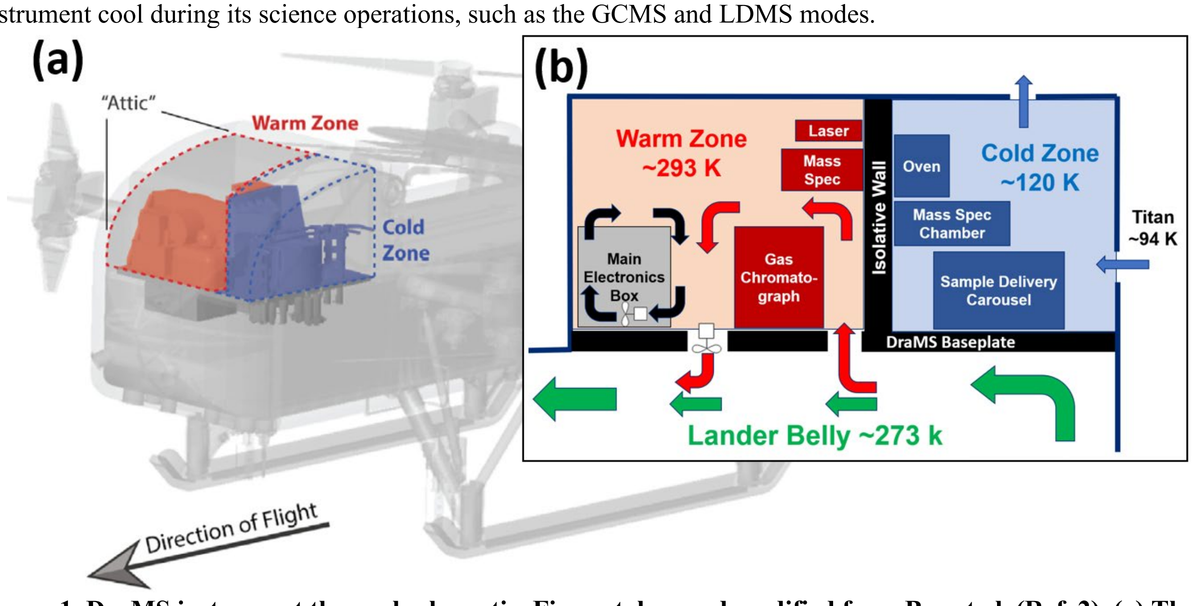


Figure 1. DraMS instrument thermal schematic. Figure taken and modified from Bae et al. (Ref. 2). (a) The warm zone houses the electronic components, and the cold zone houses the sample delivery system that stores the dug-up Titan soil samples. (b) Simplified schematic of the different temperature gas flows (represented by arrows) within the attic.

Additionally, the operational modes for DraMS are highly transient, with various components fluctuating in temperature and power significantly over a course of about 5 hours. Therefore, a robust analytical software that can handle conductive, convective, radiative, and transient heat transfer behavior is required to effectively analyze the thermal performance of the instrument. Furthermore, the software must not impose any significant restrictions and/or compromise the well-established engineering practices in the space thermal industry.

The software must be capable of mapping thermal information to structural and optical analysis platforms for the Structural Thermal Optical Performance (STOP) process. The software should not add significant complexities that would hinder model transfers with other interested parties (i.e., between DraMS and Dragonfly lander thermal teams). The software should not add significant computational cost such that the engineering design time would be significantly slowed.

Spacecraft thermal analytical software (SCTAS) such as Thermal Desktop® (TD), ESATAN, NX Thermal, THERMICA, etc., are well established in the spacecraft thermal industry and are well suited for performing comprehensive system-level conductive, radiative, and orbital analyses. While these software support simple fluid flow and convective analysis using 1-D approaches and/or empirical correlations, they do not directly solve the fluid volume conditions using Navier-Stokes (N-S) equations. When the flow field becomes complex such that the flow direction is not obvious and/or stagnant zones exist, empirical correlations and simplifications fail. To accurately capture the convective thermal behavior with SCTAS, significant testing efforts need to be done to correlate convective boundary conditions on a case-by-case basis. However, due to the nature of fluid flow, when conditions upstream change, the hard-earned correlations for the downstream components will be invalidated. On the other hand, computational fluid dynamics (CFD) software packages such as ANSYS Fluent, COMSOL Multiphysics®, STAR CCM+, OpenFOAM, etc., are great for accurately solving the flow field parameters and resulting convective conditions by discretizing the flow volume and solving the N-S equations, but they require significant computational cost. Furthermore, CFD software are usually not tailored towards large comprehensive system level modeling and their workflows are not very streamlined compared to the workflows in SCTAS.

Clear differences between SCTAS and CFD can be seen in meshing requirements and analytical workflows. For SCTAS, component model details and mesh discretization are, relatively speaking, quite independent from one another. This allows the user to easily refine or coarsen the model details of a component without affecting another, and such modularity allows a simple exchange of models and updates between design iterations. SCTAS also has many hooks to easily simulate and analyze system behaviors, such as tracking heat flows on a nodal basis, allowing quick parametrization of geometry and boundary conditions, and more. For CFD software, every component is numerically joined together via a discretized negative space (the fluid volume). This means the user must modify the entire negative space model when a design change occurs, which necessitates re-meshing of the whole system before performing additional analysis. This further complicates model exchanges if the fluid volume is shared between multiple parties. CFD also requires fine meshing near boundary layers to accurately capture near-wall effects, which requires more of the analyst's time for meshing checks and more computational time. Finally, CFD models suffer significant computational time penalties when transient analysis is involved due to the strict numerical stability requirements noted by the Courant-Friedrichs-Lewy (CFL) condition, which defines the maximum allowable fluid information travel per solver time step (i.e., fluid velocity multiplied by the time step) as a ratio of the mesh size in the direction of flow travel.

This paper investigates a method to join a CFD software with an SCTAS to aid the engineering development efforts and the thermal analysis of the complex thermal behavior of the DraMS instrument. Specifically, this paper investigates co-solving of ANSYS Fluent and TD due to their recent publication[3] and feature development. Note, however, the benefit of co-solving is not limited to TD-Fluent only, and can be extended to other software, should the software support external communication with their solvers. Additionally, this paper proposes a methodology called the Thermal Iso-value Exchange (TIE) method to further improve the workflow of the TD-Fluent co-solver for the DraMS instrument.

II. DraMS Thermal Analysis Before the Co-Solver

Before the implementation of TD-Fluent co-solver, the thermal behavior of the DraMS instrument had been analyzed using only TD. The warm zone of the DraMS instrument has two compartments: the Gas Chromatograph (GC) compartment, and the main compartment. Just two air nodes were used to represent the reference air temperature for the components, with the assumption that the air is well-mixed within each of the compartments. Figure 2 shows the compartment locations. Note that cold attic is sealed from fluid flow exchange with the two compartments and is uninteresting from the perspective of the co-solver, so it will not be discussed further in this paper.

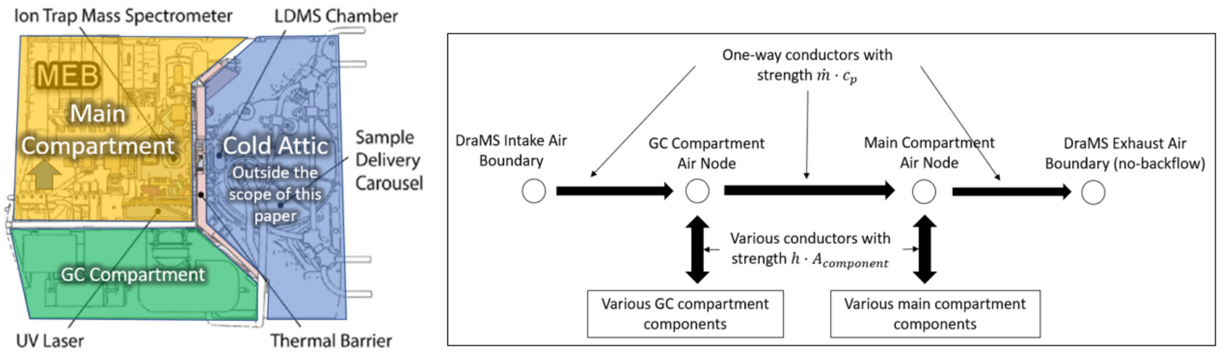


Figure 2. DraMS attic zone partition from the top-down view and a simple 1-D method used for simulating convective conditions in the DraMS TD model before co-solver implementation.

Figure 2 shows how the simple 1-D conductors were used to simulate the gas flow conditions in DraMS. While this method was very simple to implement and allowed quick thermal analysis, there were significant downsides. The 1-D method assumed that the air was well thermally mixed within each of the compartments. The convective strength, while informed from empirical correlations, was assumed to be uniform per component because local variations due to developing length could not be estimated without CFD. However, as shown via the CFD modeling of DraMS in Figure 3, the flow profile inside DraMS is very complex. There are significant local air velocity variations, which would cause significant h value variations. Furthermore, there are large local air temperature variations, which challenges the well-mixed assumption used in the 1-D flow analysis.

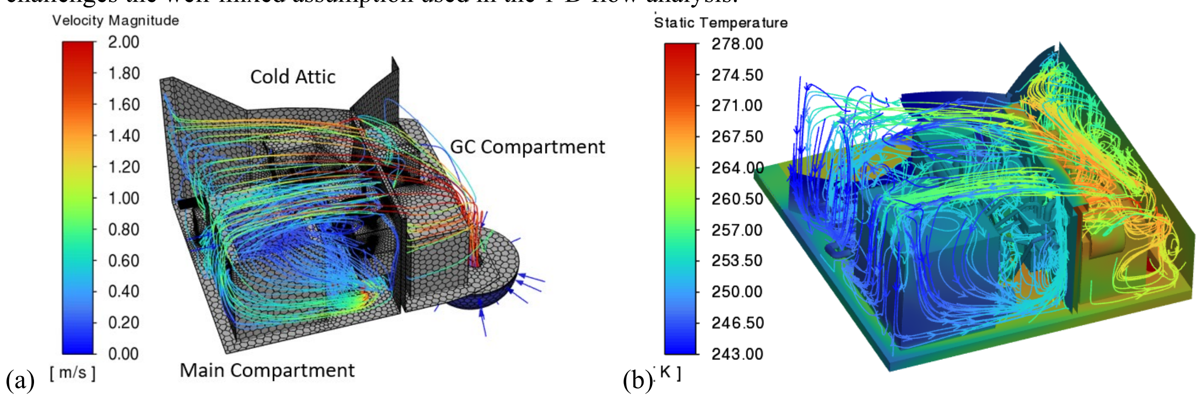


Figure 3. An isometric view of DraMS attic with Fluent velocity path-lines showing the complex flow pattern (a) and the temperature contour (b) inside the DraMS warm zone.

To accurately capture the thermal behavior inside DraMS, CFD modeling was necessary. As opposed to building the entire system level thermal model within a CFD software, the co-solver feature was chosen as the best path forward. Co-solving allows analysts to continue using the TD thermal model and add on the CFD to inform the TD system level model instead of using the rudimentary 1-D method.

III. TD-Fluent Co-Solver Investigation

The TD-Fluent co-solver process is explained in detail in Ref. 3 and Ref. 4. Therefore, only a summary is provided here. The co-solver utilizes ANSYS System Coupling (SyC) software to specify and create a data transfer medium. The co-solver offers various data transfer methods, but this paper mainly focuses on the Q-T method. The Q-T method works as follows, and is visualized in Figure 4:

1. TD provides temperature information (T_{surf}) to SyC.
2. SyC associates TD elements to Fluent elements to map and forward temperature information to Fluent.
3. Fluent calculates the local surface heat fluxes ($\dot{Q}_{conv}$) by holding T_{surf} constant.
4. Fluent sends the converged heat flux values to TD via SyC.

5. SyC uses the associations established in step 2 to map and forward heat flux information to TD.
6. TD uses the heat flux values as additional boundary conditions in the model to solve for the new T_{surf} values.
7. Once TD solution is converged, process repeats until SyC no longer sees significant $\dot{Q}_{conv}$ and T_{surf} changes.

It is important to note that the user-defined co-solving surfaces in TD and Fluent are aligned properly with one another because the data injection is handled via an automated mapper[4]. This allows the system level model to stay in TD, where system level dynamic conditions can be easily modeled and analyzed, while Fluent can inform the system level model of the complex interactions between all the components.

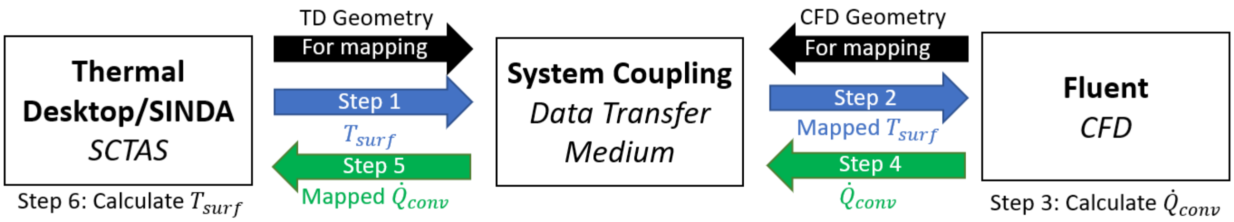


Figure 4. Simplified TD-Fluent co-solver data transfer process

The following attributes of the co-solver were investigated:

A. Numerical accuracy: does co-solving produce any additional numerical error?

B. Numerical stability: Does co-solving produce any stability or convergence issues?

C. Complexity: How disruptive is the co-solver process to the standard practices in Fluent and TD model?

D. Transient capability: Can the co-solver handle transient thermal problems?

E. Correlation and uncertainty biasing capability: Can the user easily modify the model/algorithm to match the test data or apply uncertainty factors easily?

The following subsections will discuss each of these questions. A summary of findings is presented in Table 2.

A. Numerical Accuracy

To test the numerical accuracy, a simple laminar flow over an isothermal and isoflux plate was used for comparison. Well-established empirical correlations, CFD only result, and co-solved result are compared. Well-established empirical correlations are used to validate the quality of the baseline CFD model. The results are shown in Table 1. Dimensions of the flat plate are 10 cm long and is infinitely wide (for net heat flow calculations, 0.1 m width section is used). Flow conditions are 293 K uniform 0.1 m/s air (ρ=1.225 kg/m3, k=0.0242 W/m·K, c_p =1006 J/kg·K, μ=1.789·10^{-5} kg/m·s). With these conditions, the maximum Re_x≈700, which is well below the critical Reynolds number of around $5 \cdot 10^5$ (Ref. 5), so laminar flow can be assumed. The flow problem is visually shown in Figure 5. Fluent simulation shows good agreement (slight differences might be attributed to edge effects or empirical correlations not being perfect), and co-solved simulation shows the exact same result as the Fluent only simulation. In the non-isothermal case, slight total heat flux difference of about 3% between Fluent only and co-solved simulations were noted, which showed that the temperature mapping process might have contributed to the differences. Visual inspections of Fluent only vs co-solved simulations yielded no significant temperature profile differences. Given the small difference (<5%) and no major findings from the visual inspections, the differences were deemed acceptable.

Table 1. Comparison of Empirical vs CFD only vs TD-Fluent Co-Solved Results

Simulation Case	**Empirical** *Eq. from Bergman et al.*[5]	**Fluent Only Simulation**	**Co-Solved Simulation** *TD provides temperature, Fluent provides convective heat flows*
Isothermal $T_{surf} = 303K$	$\overline{Nu}_x = 0.664 \cdot Re_x^{\frac{1}{2}} \cdot Pr^{\frac{1}{3}} = 15.7 \rightarrow \dot{Q}\big\|_{w=0.1[m]} = 0.38$ W	$\dot{Q}_{Fluent} = 0.391$ W	$\dot{Q}_{TD} = 0.391$ W $\dot{Q}_{Fluent} = 0.391$ W
Linear Temperature Profile $T_{surf} = 303K \rightarrow 293K$ *from leading to trailing edge*	--	$\dot{Q}_{Fluent} = 0.176$ W	$\dot{Q}_{TD} = 0.171$ W $\dot{Q}_{Fluent} = 0.171$ W

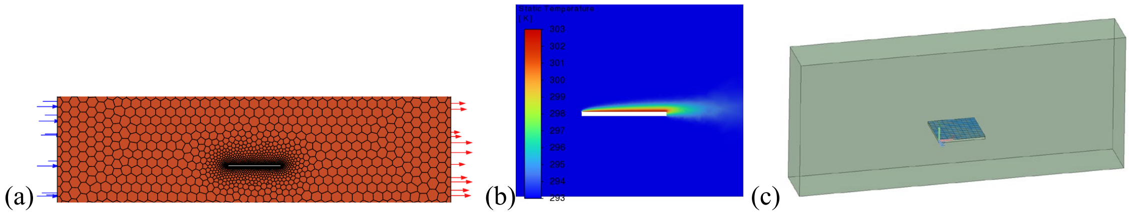


Figure 5. (a) CFD Mesh of the plate flow problem used for both Fluent only simulation and co-solved simulation, (b) the temperature contour of the Fluent-only result for the isothermal case, and (c) Isothermal TD model overlaid with the transparent fluid geometry model from Fluent. TD model is simply just the plate.

However, Fluent and TD yielded notable differences in local convective heat flux distribution and magnitude. Further investigation revealed that the local flux information gets smeared during the data transfer process (step 4 in Figure 4) due to the differences in the mesh discretization between the two software packages and due to the differences in the control volume implementations of TD (nodal-based) and Fluent (element-based). Currently (version 2023R2), the co-solver maps the flux information directly from a Fluent element to a TD element, whereas the information should be mapped directly to the TD nodal volume to be consistent with the TD solver. This inconsistency can cause errors locally on the convection participating surfaces if the local convective flux variations are high (i.e., at the developing regions of the flow), if the local Biot number is high, and/or if the nodalization is too coarse on the TD side (which effectively dictates the local/nodal Biot number). With sufficiently low local Biot number, the surface temperature can become insensitive to the variations in the local heat flux. However, at the opposite extreme, this could lead to a development of an unphysical surface temperature profile. For DraMS, since there are many insulation materials, such as Rohacell® 31 ($k \approx 0.025$ W/m·K), participating in convection, process improvement was needed to mitigate this problem. Figure 6 shows an example of an extreme case where improperly mapping the across the control volume can yield unrealistic and unphysical solutions.

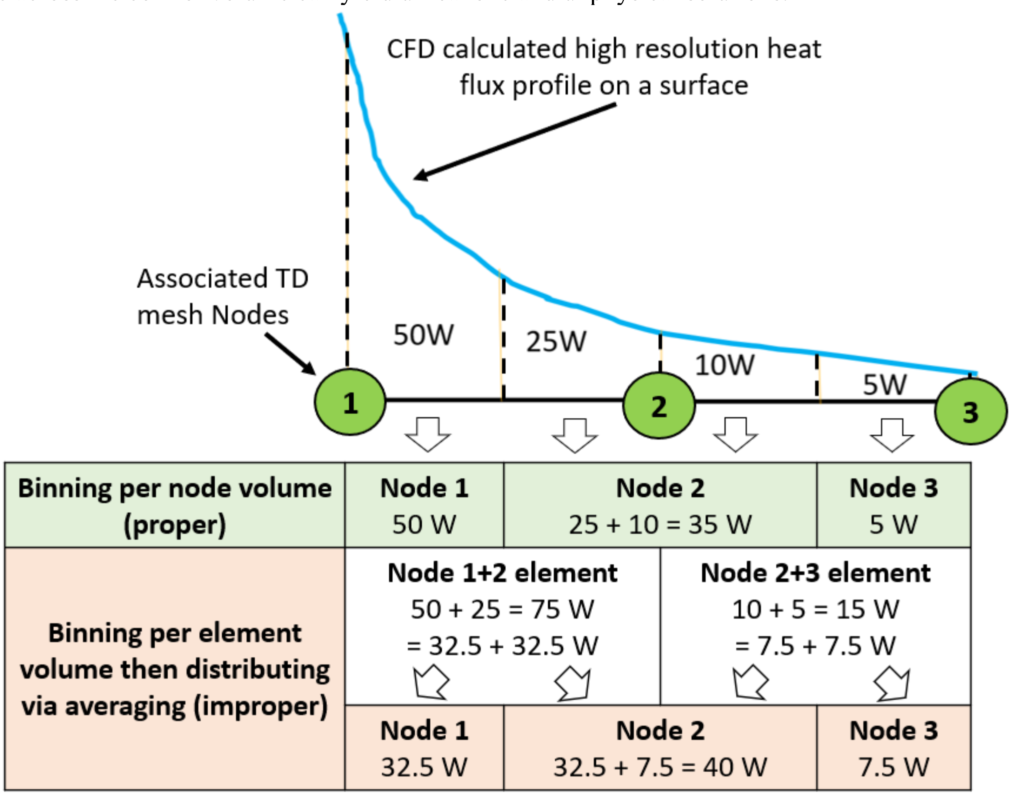


Figure 6. A figure depicting how improper binning/mapping of the heat flow at the interface with large mesh discrepancy between two communicating software can lead to large errors and unphysical heat flux distribution.

Note how with the improper mapping, the middle node has the highest heat flux which can lead to unphysical results. Figure 7(a) replicates the issue presented in Figure 6 by simulating hot airflow over an isolative ($k = 1$W/m·K, thickness = 1mm, 1m x 1m) floating plate. The solution is unphysical because the middle of the plate is hotter than the heat source (air) temperature of 293K.

While increasing nodalization fixes the issue, as shown in Figure 7(b), there is a desire for analysts keep a low node count to reduce the radiative computational costs and for model exchanges. Therefore, proper binning and mapping of heat fluxes to correct control volume is desired.

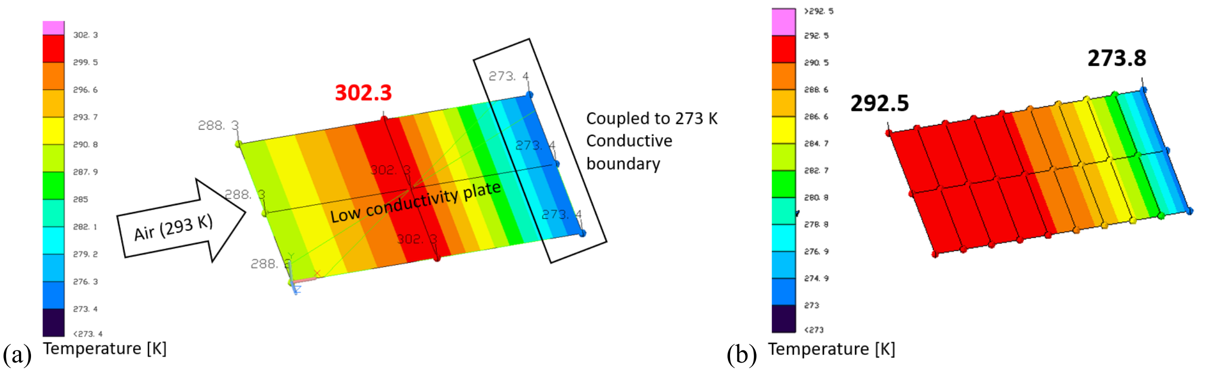


Figure 7. (a) shows an unphysical result (surface is hotter than the heat source) arising from improper temperature mapping for an extreme case where the local Biot number is extremely high. (b) shows how the increased mesh density fixes the issue by effectively increasing the local nodal Biot number and discretizing the energy bins into smaller chunks.

B. Numerical Stability

The HT-T co-solver method is worth discussing for numerical stability. Instead of supplying $\dot{Q}_{conv}$ at the data transfer step from Fluent to TD (steps 4 and 5 from Figure 4), the HT-T method instead supplies the Fluent calculated local convective heat transfer coefficient, h, and the accommodating local reference air temperature, T_{ref}. The HT-T method is superior to the Q-T method when it comes to numerical stability because the imposed boundary conditions force the solution to be bounded. Consider a sample problem where a plate is floating in air such that the plate temperature should follow the air temperature, and the parameter of interest is the steady state temperature of the plate. While the solution is trivial for the HT-T method where $T_{surf} = T_{air}$, for the Q-T method, the problem is unbounded because the air temperature T_{ref} is lost during the data transit. Since the floating plate is poorly thermally coupled to any other boundary conditions (or not at all) for such case, the plate temperature will either skyrocket or plumet, unbounded, depending on whether the initial guess was below or above the air temperature, respectively. Figure 8 shows the temperature of a co-solving participating floating plate with the HT-T method versus the Q-T method.

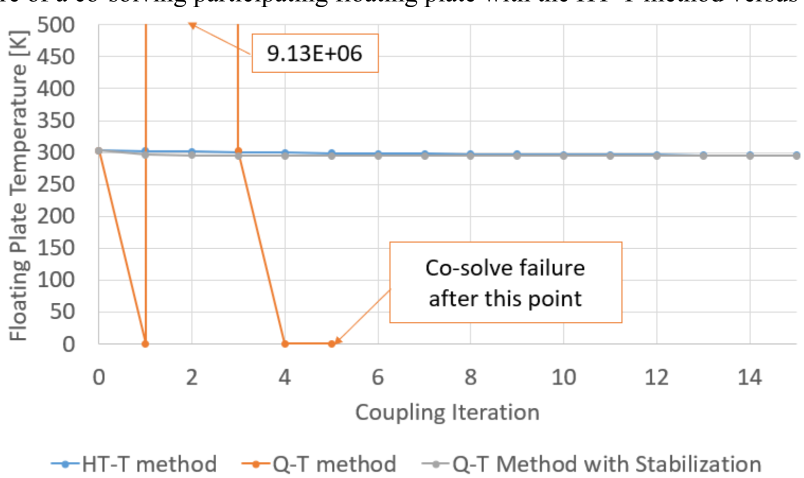


Figure 8. Temperature vs. co-solving iteration for the HT-T method and the Q-T method for a floating plate problem

The Q-T method with stabilization uses a hybrid approach, where a temporary heat transfer coefficient with an estimated reference temperature is applied during the solver routine to stabilize the temperature node (i.e. $\dot{Q}_{Total} = C \cdot \dot{Q}_{Fluent} + (1 - C) \cdot \left(h \cdot A \cdot \left(T_{surf} - T_{ref}\right)\right)$ where C is a parametrizable constant to help with numerical stability. As can be seen by the "Q-T Method with stabilization" curve in Figure 8, this method showed great stability and showed the importance of applying proper stabilization technique. Note that due to the control volume issue discussed in the Numerical Accuracy subsection, HT-T method was unable to conserve the total net convective heat flow during the data transfer unlike the Q-T method. Since net energy conservation was important, Q-T method with stabilization algorithm was chosen.

C. Work-flow Process

For the co-solver to be useful, its workflow process cannot be too disruptive. Many deep space missions have tight launch windows, and analysis times must be quick enough to meet tight engineering deadlines. The co-solver utilizes data mapping to transfer data to and from TD and Fluent to allow easy association between the models. For the data mapping to work properly, the geometry must be similar enough for data mapping to be successful. However, when working with radiative models versus convective models, the model defeaturing process can be significantly different. For a radiative model, ribs and surface features on a surface can be mostly ignored since effective emissivity can easily take care of the added area. However, for a convective model, the surface features can add significant convective area and affect the flow downstream significantly, so the convective model will require more modeling fidelity. For a convective model, negative area is just as important as the component model itself because the mesh fidelity of the fluid dictates how accurately N-S equations can resolve the fluid flow and near wall effects. However, for a space thermal model, negative space is insignificant and model defeaturing process can be more flexible. Figure 9 shows one such example of where convective model might have a problem whereas a space thermal model would not. For the shown case, the CFD model might require a cutaway a section of the cylinder to allow the gap to be bigger so that meshing density would not increase as much.

Furthermore, only the finite element type surfaces are supported in the data transfer between TD and Fluent currently (version 2023R2), and the DraMS TD model implements many non-finite element surface types. Because of the different defeaturing requirements between a TD model and a Fluent model and additional modeling restrictions, setting up a model for co-solver is an added challenge to the modeler.

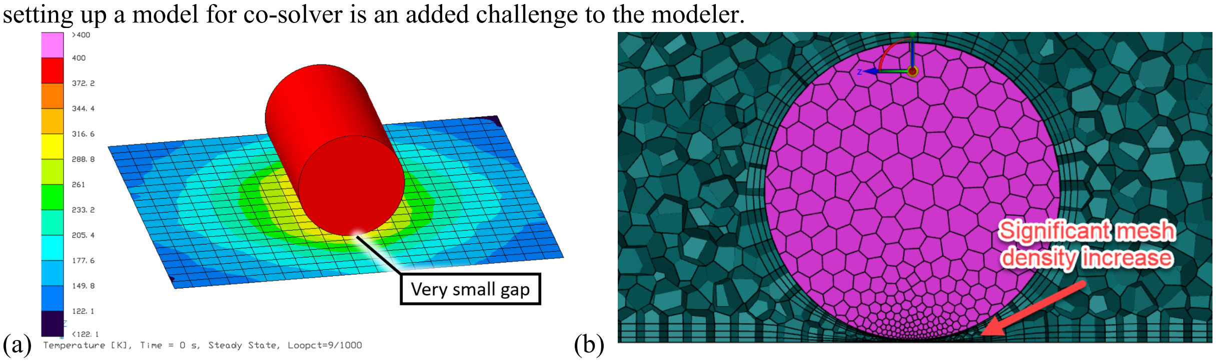


Figure 9. TD model (a) vs Fluent model (b) when working with a close junction between components.

D. Transient Simulation

The co-solver allows transient simulation using quasi-transient method, where the TD solves the nodal network in transient while Fluent performs only steady state simulations. Fluent solves steady state N-S equations using the temperature snapshot taken from TD at each coupling time step. Such quasi-transient method is valid because viscous flow time constants are significantly smaller than thermal time constants for gas flows (i.e., Pr is small). In other words, viscous flow profile will be steady long before thermal flow profile becomes steady, assuming thermal changes do not significantly impact the viscous profile. Furthermore, the thermal capacitance of gas is significantly lower than the thermal capacitances of many DraMS components, so the thermal transient behavior of the gas itself can be ignored. This is a huge benefit for co-solving; since CFD portion can be steady state, time terms can be taken out of the N-S equations and the maximal allowable timestep restriction from the CFL condition becomes irrelevant.

The co-solver allows the coupling steps to occur less frequently than the time steps from TD, which is highly desirable for computational time savings. In this case, TD would hold the Fluent provided convective boundary condition values constant until the next coupling iteration occurs. If the coupling frequency is too low and/or the application of the convective boundary condition is improper, this can cause errors in the transient response. Figure 10 shows how the transient response can vary with the coupling iteration frequency and with different approaches to applying the convective condition. Note that the Q-T quasi-transient method will always yield faster temperature approach rate than reality (which was simulated with h = 20 W/m^2·K) if the coupling time-steps are less frequent than the TD model because there is no feedback to decay the convective heat flow in-between the coupling interval. The HT-T method will mimic the reality better, but it will either overshoot or undershoot depending on whether the h value estimated by the N-S equation is lower or higher than the actual h value, respectively. Ultimately, when using the quasi-transient method, it is recommended to perform a coupling interval frequency independence study to see if the chosen coupling interval frequency is sufficient. Furthermore, it would be beneficial to convert the Q-T method to HT-T method so that the decaying behavior can be captured, as it will be discussed in the TIE method section.

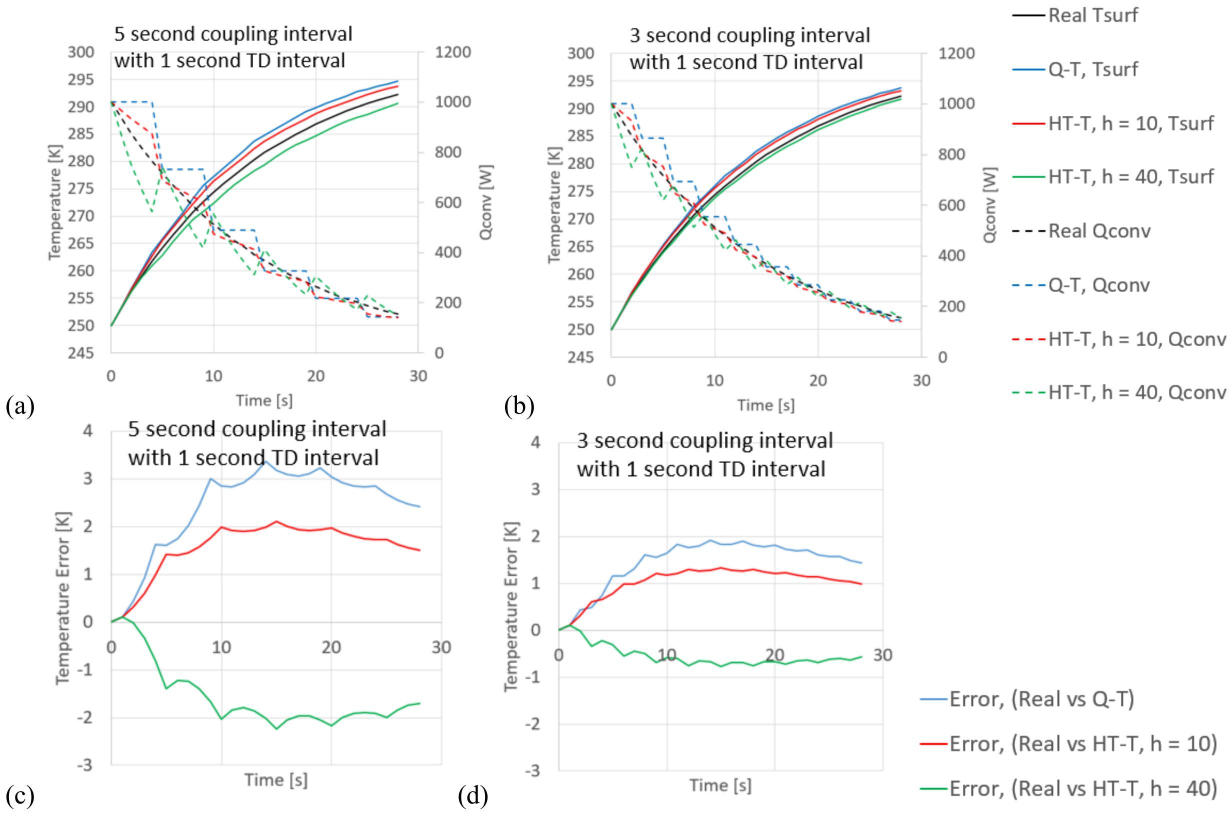


Figure 10. Floating plate problem results with intermittent co-solving frequency. (a) and (b) show the convective heat loss (dashed lines) and temperature response (solid lines) versus time with 5 and 3 second coupling intervals, respectively. (c) and (d) show the temperature deviation from real with 5 and 3 second coupling intervals, respectively.

E. Correlation and Uncertainty Biasing capability

Model correlation via testing is a major effort, but necessary part of any space flight components. Unfortunately, directly correlating a CFD model is a difficult task due to how upstream components can affect downstream components. Furthermore, since the convective conditions are dictated by N-S equations which depend on many complex variables such as geometry shape, correlating a CFD model to match the test data can be a difficult task. However, the data transfer medium via the co-solver exposes the convective heat flow data to the user, which gives the user direct control over the convective data. For example, for the Q-T method, the heat flows from the CFD can be either amplified or subdued at the user's will, which the user might want to do to add some uncertainty values from the CFD model or tweak to correlate to a test data. Table 2 summarizes the key finding in each of the attributes that was of interest for DraMS thermal modeling.

Table 2. Summary of the TD-Fluent Co-Solver Investigation

Attribute	Summary of Findings
Numerical Accuracy	TD to Fluent temperature data transfer is accurate. Q-T method conserves net energy flow. Local energy flow information gets smeared due to control volume discrepancy and mapping issue between Fluent and TD.
Numerical Stability	HT-T method is inherently more stable than Q-T method. Q-T method requires stability control algorithm (i.e., mimicking the HT-T method) for convection dominant problems.
Work-Flow Process	Data mapping automatically handles TD-Fluent surface associations. Defeaturing process differences between radiative vs convective models and unsupported surface types in the co-solver pose challenges for modeling and data mapping.
Transient Simulation	Co-solver utilizes quasi-transient method, where CFD is run in steady state with the assumption that the viscous time constant is significantly lower than the thermal time constant. This allows much larger time steps to be taken for transient simulation compared to CFD only transient simulation.
Biasing Capability	Co-solver exposes convective boundary condition data and allows user to freely modify at will, making convection correlation easier.

IV. The TIE Method

A data handling method called the Thermal Iso-value Exchange Method (TIE) method was developed to tackle the challenges in the current implementation of the TD-Fluent co-solver. The concept of the method is very simple and follows very similar data transfer process described in Figure 4; the only difference is that the TIE method interjects a surrogate co-solving model between the original TD model and SyC to handle data communications. The surrogate TD model is more akin to the CFD model, and simply exists as a data mapping and handling medium. The surrogate model does not contribute to the radiation data exchange nor conduction calculations. The surrogate model isothermalizes the temperature data from the original TD model via associations with user-defined domain tagsets (surface groupings). The isothermal data gets sent to Fluent, which will resolve the flow profile using the isothermal data. Then Fluent sends the resulting convective heat flow information ($\dot{Q}_{conv}$) back to the surrogate. The surrogate model then converts the heat flows ($\dot{Q}_{conv}$) into reasonable T_{ref} and h combinations, then applies them to the original TD model as iso-convective boundary condition (i.e., no h variation between node to node). Figure 11 visually shows how the TIE method works.

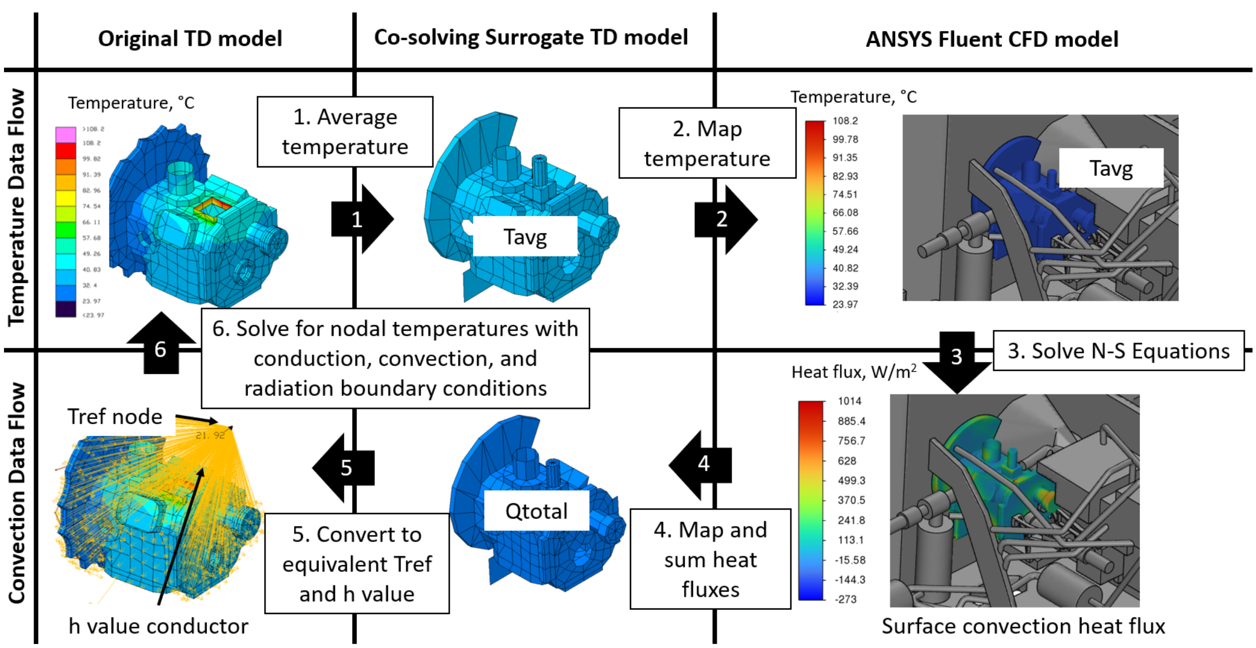


Figure 11: TIE method data flow overview

There are several benefits to the TIE method over the direct co-solver method. First, it decouples the original TD model completely from the CFD model; the TD model can look significantly different from the CFD counterpart. For example, the original TD can be a simple box as opposed to a ribbed and finned box. This means that the model defeaturing can be done without any regard to what the CFD model needs, allowing the CFD modeler to work independently from the TD modeler and vice versa. This further allows users to continuously use non-finite element surfaces for the original TD model, since the finite element requirement now only needs to be applied on the surrogate model. This additional geometric layer provides an additional buffer that prevents any possible disruptions for the CFD and the TD modelers and can improve data mapping between the TD and Fluent because the surrogate model can be an exact match to the CFD model without affecting the TD model. Because the surrogate does not participate in radiation or conduction, the surrogate model can be as detailed as needed without impacting computational time.

The TIE method also gives a user easier access to the data transfer. When the heat flux data from Fluent is converted to h and T_{ref} values (happens in Variables 0 in TD), h values could be biased higher or lower. The analyst can utilize this control to apply biasing for hot/cold analysis and/or adjust to match test data without having to modify the corresponding CFD model.

The TIE method does not suffer significantly when it comes to numerical accuracy. The method utilizes the Q-T method, meaning the net exchange is conserved per the investigation findings. The control volume issue at data exchange is no longer an issue because all the local convective data is forcibly removed via the isothermalization and iso-convection steps. This, of course, will deviate the thermal model from the true physical solution if the surrogate handles too large of an area, but this is still a more accurate solution than the 1-D convection implementation for DraMS as shown in Figure 2. The DraMS 1-D approach cannot capture the local variations in the fluid conditions, whereas the TIE method can still capture the variations in the fluid via the usage of CFD. Furthermore, should the analyst wish to capture local convection fidelity better, the user can partition the surrogate to be smaller to better capture local variations.

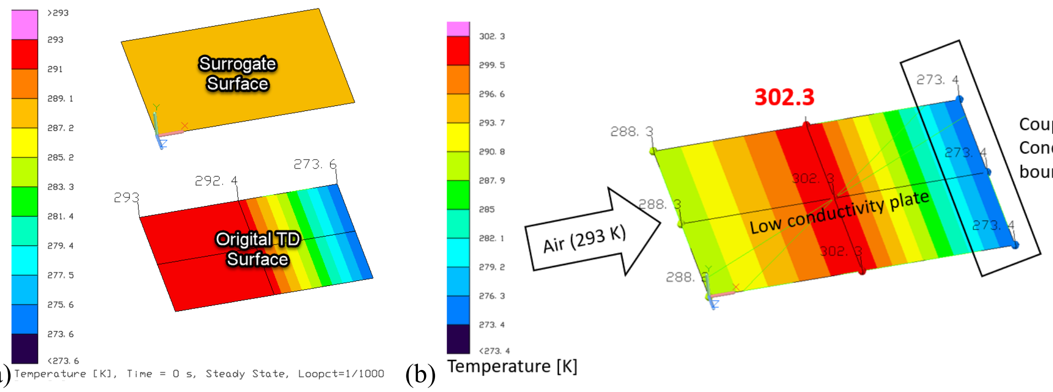
The TIE method also offers robust stability by changing the Q-T method to pseudo-HT-T method when it reapplies the convective condition back to the original TD model. The h-T_{ref} combination is calculated as follows:

1) Assume small but reasonable (i.e., use empirical correlations) h value per surrogate surface: h_i where i represents surrogate surface id.
2) Solve for $T_{ref_i} = \frac{Q_{conv_i}}{h_i \cdot A_{surf_i} \cdot T_{surf_i}}$
    a. Note that the implementation avoided solving for h with assumed T_{ref} because doing so can cause divide by zero error if, by chance, T_{surf} is equal to T_{ref}.
3) Limit $\left|T_{ref_i} - T_{surf_i}\right| < \Delta T_{max}$ where ΔT_{max} is a user specified variable, but small enough such that T_{ref_i} can never be below absolute zero. Note that T_{ref} is representing average near wall temperature, which should never deviate too much from the surface temperature, so to mimic physical behavior, a reasonable limit is set.
    a. If $\left|T_{ref_i} - T_{surf_i}\right| > \Delta T_{max}$, set $T_{ref_i} = T_{surf_i} \pm \Delta T_{max}$ ($\pm$ depending on heat flow direction), then calculate for new h to satisfy the energy conservation: $h_i = \frac{Q_{conv_i}}{A_i \cdot \Delta T_{max}}$, which will be used for all subsequent co-solving iterations for that particular surrogate surface until the limit is violated again.

Note that a very high initial h value guess can lead to significant problems during transient solutions and cause the problem to be too convection dominant even when the problem is not supposed to be. For example, during a transient simulation, if the co-solving step occurs once every 10 SINDA timesteps, too high of an h value might cause the T_{surf} to approach the T_{ref} value too quickly. This means that the transient convective response will be dampened out unless the user makes the co-solving step occur more frequently (see h=40 W/m^{2}K curves in Figure 10). Furthermore, since a single bulk near-wall temperature is used, too high of an h value can lead to unintentional temperature smearing, and at an extreme, the entire associated surface will simply equal the T_{ref} due to h value being too strong. Therefore, it is beneficial to initially guess a small h value, then adjust it to be higher as needed. However, the authors acknowledge that there is room for improvement in the correct h value choosing.

As for the workflow, the TIE method improves the workflow process by modularizing the co-solving aspect of the model. The users could very easily switch back and forth between the co-solving and non-cosolving model runs by attaching and detaching the surrogate model. The additional layer of separation also allows users to easily exchange TD only model independently. However, the TIE method comes at the cost of additional setup time. Additional groupings (domain tagsets in TD) must be created to associate the surrogate surfaces to the original TD model. However, the OpenTD API[6] can be utilized to automate this process.

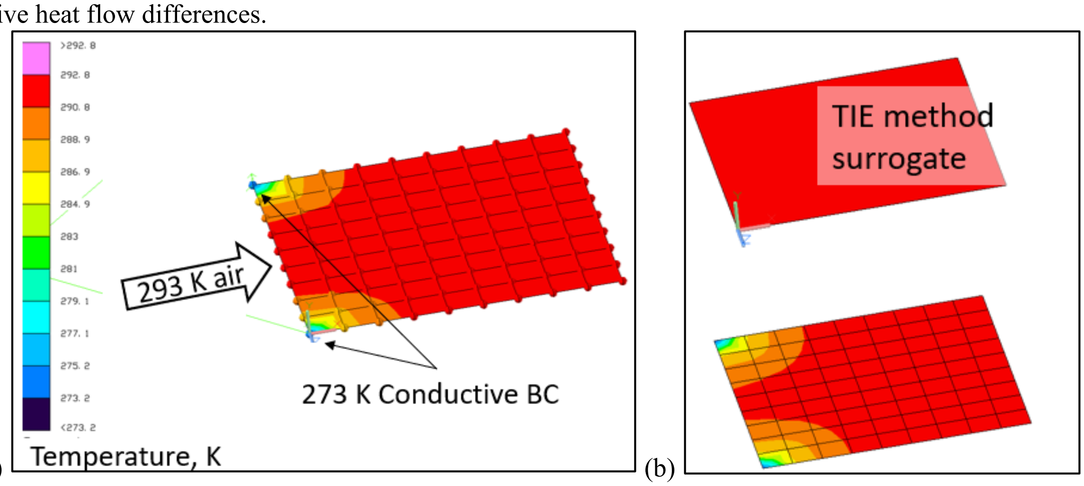
Figure 12 shows the numerical robustness of the TIE method over the direct TD-Fluent co-solver by showing how the TIE method can resolve the problematic high Biot number problem shown in Figure 7 without having to increase the mesh discretization.

(a) 

(b)

Figure 12. (a) The floating problem result with a very high local Biot number solved using the TIE method. (b) Figure 7(a) is reshown for comparison.

Figure 13 shows the result from the TIE method versus the result from the direct mapping method offered by the TD-Fluent co-solver for a more realistic case, where local Biot number is not forced to be extremely high. The plate is a very low thermal conductivity material to enforce large temperature gradient locally. As expected, the TIE method results in smoothed out temperature compared to the true solution because the local convection information is lost in the TIE process. In other words, if the TD model already has sufficient mesh discretization to resolve all the local heat flux variations and temperature gradients, the original TD-Fluent co-solver will yield more accurate result than the TIE method can. However, since the TIE method is more tolerant of the mesh size than the original implementation of the co-solver, the TIE method is still a desirable implementation. Additionally, if higher accuracy is desired using the TIE method, the surrogate surface can be partitioned around the high gradient locations to better capture the local convective heat flow differences.

(a) 

(b)

Figure 13. Comparison between direct TD-Fluent co-solver (a) vs. the TIE method (b) for a well nodalized/low local Biot number surface.

Furthermore, the TIE method resulted in faster convergence rate (4 coupled iterations) than the original method (15 coupled iterations) for the example shown in Figure 13, which makes sense considering the isothermalization and iso-convection steps should smooth out any significant local numerical instabilities that might occur between iterations. And, as discussed in Figure 10, TIE method improves the transient response co-solving compared to the direct Q-T method because the TIE method converts the Q-T method into HT-T method. Considering modularity, better convergence times, and improvements in convection simulation accuracy over the 1-D method, the TIE method was chosen for the DraMS thermal model. The following section discusses the DraMS model results with the TIE method implementation.

V. DraMS Thermal Analysis with the TIE Method

Figure 14 shows the DraMS TD model, its TIE co-solving surrogate, and the corresponding CFD model. A total of 57 surrogate surfaces was created to transfer the data. Note how the surrogate and the CFD model shows how the temperature was partitioned into different iso-value zones.

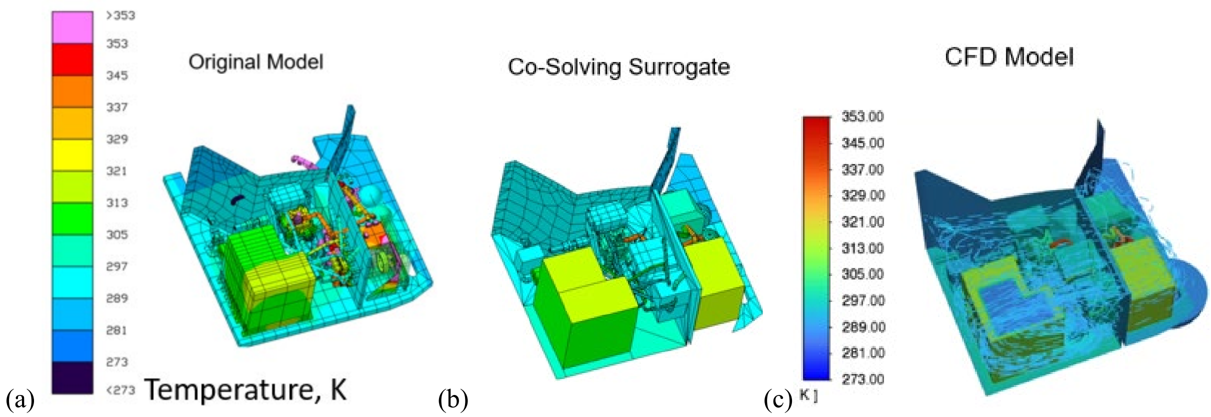


Figure 14. DraMS TD Model (a), co-solving surrogate model (B) and the corresponding CFD model (C), which is all used for the TIE method.

The steady state model with co-solver using the TIE method took about 75 minutes as opposed to TD only model that took about 15 minutes. For a 5-hour transient simulation, the co-solver with the TIE method took about 9.5 hours to compute as opposed to TD only model that took about 6 hours. The heat flow comparison between the co-solved run vs. non-co-solved runs are shown in Table 3. For the 1-D method, the convection coefficient was varied $\pm$ 50% (10 $\pm$ 5 W/m^{2}K) to account for the uncertainties in the local air velocity, local air temperature, surface orientation, and geometry differences between components. Despite the large biasing, the co-solver has shown that several components were still losing or gaining more than expected, which shows that utilizing CFD is very important to accurately capture the convective strengths. Note that the default co-solving approach was not performed for the DraMS thermal model because of the numerical accuracy and workflow limitations discussed in Section III.

Table 3. Heat flow comparison of the DraMS thermal model results with and without the TIE co-solver

Convective Heat Flows, $\dot{Q}_{conv}$. Positive is heat into the air			
DraMS Component	**TIE method Co-solve**	**1-D method h = 5 W/m^2·K**	**1-D method, h = 15 W/m^2·K**
Attic Baseplate	6.8	3.82	5.26
Attic Roof Structure	-37.5	-32.7	-46.5
Detector	1.49	1.29	1.43
Ion Trap Mass Spectrometer (ITMS)	4.69	3.63	4.83
Laser	0.55	0.51	0.68
Manifold Valve Assemblies	0.036	0.55	0.38
Manifold Support Wall	3.27	0.64	1.17
Miniature Scroll Pumps (MSPs)	4.84	3.22	6.82
RF Electronic	1.74	0.64	1.57
Wide Range Pump	4.19	1.85	3.72
Wonderwall (structural)	-3.72	-4.92	-8.78

Finally, Figure 15 compares the transient temperature response of some of the DraMS instrument components during the hot biased GCMS mode with and without the TIE co-solver. Some components, like the MSPs, show very similar behavior between the co-solved versus the TD only model, which indicates that either the TD model with 1-D convection approximation was good, or that the component is not convection sensitive. Some components, like the detector, show significant variation between the two, indicating that 1-D method was not sufficient to accurately capture the convective losses. It is interesting to note that the detector shows similar sawtooth behavior as shown in Figure 10, which might indicate that the *h* value assumption for the detector was either set too high or there were some flow instabilities present in the CFD model. Overall, the co-solved model showed generally cooler component predicts than the TD only model, which showed that the 1-D guesses for the *h* values were too conservatively low.

With the successful DraMS co-solved model using the TIE method, the authors believe that the co-solver and the TIE method is scalable to large scale system level thermal models.

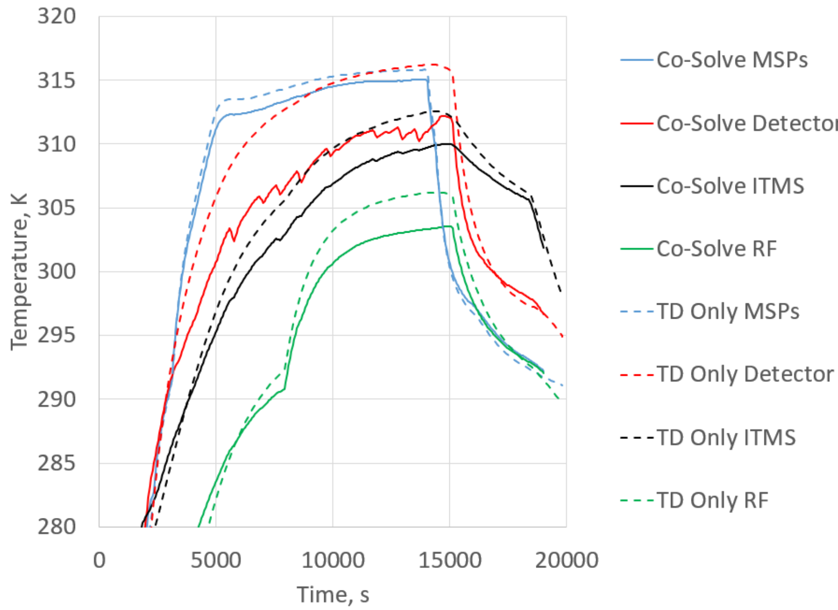


Figure 15. Transient temperature prediction of some DraMS instrument components for the hot biased GCMS mode. The solid lines represent co-solved results using the TIE method and the dashed lines represent TD only results using the 1-D convection method described in Figure 2.

VI. Conclusions and Future Work

Spacecraft thermal modeling software with computational fluid dynamics software co-solving was investigated for the thermal analysis of DraMS instrument, which requires proper handling of conduction, convection, and radiative heat transfer modes. TD-Fluent co-solver was chosen, and a methodology called the TIE method has been proposed to improve the co-solver process in terms of workflow and data handling. Sample results from the DraMS instrument are used to show the feasibility and scalability of the co-solver and the TIE method. Results show that co-solving is crucial for better engineering of the DraMS instrument and will be continuously used for DraMS development. Notably, the convection data adjustability capability with the TIE method will be helpful for any future test data correlation activities.

Co-solver methods are not perfect as the feature is still in its infancy. Setup and implementation workflows have room for improvement, and care must be taken when transferring data between software with fundamental differences in control volumes, as it can lead to accuracy issues during data transfer. The author believes that the co-solver and the TIE method have significant room for improvement, such as the implementation of finding a better *h* value to avoid issues with data smearing via convection coupling and improvement upon transient data handling.

Acknowledgments

The authors would like to thank the DraMS project and NASA GSFC MRAD for funding this effort and the various project personnel on the support of this activity. Furthermore, the authors would like to thank Tim Panczak from the Thermal Desktop® team from ANSYS for providing insight into the co-solver process.

References

[1]Hautaluoma G, Johnson A., "NASA's Dragonfly Will Fly Around Titan Looking for Origins, Signs of Life," URL: https://www.nasa.gov/press-release/nasas-dragonfly-will-fly-around-titan-looking-for-origins-signs-of-life [cited 6 April 2023]

[2]Bae, D., Steinfeld, D., Robinson, F., and Nichols, S., "Thermal Design and Control of the Main Electronic Box in Titan Environment for the DraMS Instrument," *52nd International Conference on Environmental Systems*, Calgary, Canada, 2023.

[3]Panczak, T., "Co-solving 3D Fluid Flow and Heat Transfer with CRTech Thermal Desktop and ANSYS Fluent," *Thermal and Fluids Analysis Workshop 2020*, Virtual, URL: https://tfaws.nasa.gov/wp-content/uploads/TFAWS2020-CRTech-CHT.pdf [cited 30 April 2024]

[4]"Coupled Heat Transfer and Fluid Flow with Thermal Desktop and Ansys CFD for Release 2023R2," ANSYS, 2023

[5]Bergman, T.L., Lavine, A.S., Incropera, F.P., Dewitt, D.P, *Fundamentals of Heat and Mass Transfer*, 7th ed., John Wiley & Sons, Chichester, England, 2011, pp. 442-446

[6]Garrett, M. D., Panczak, T. D., Schmidt, M. J., and Wilkins, D., "Getting Started with Open TD Version 2023 R2," 2023.