

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF OPTICAL PAYLOAD FOR LASERCOMM SCIENCE (OPALS) SEALED ENCLOSURE MODULE

Dr. Kevin R. Anderson, P.E.*, Daniel Zayas, Daniel Turner*****

*NASA Jet Propulsion Laboratory (JPL) California Institute of Technology (Caltech)
Thermal & Fluid Systems Engineering Group, Faculty Part Time, CFD Fluids/Analysis
Engineer, Professor of Mechanical Engineering, California State Polytechnic University at
Pomona

*NASA Jet Propulsion Laboratory (JPL) California Institute of Technology (Caltech)
Thermal Cognizant Engineer OPALS Project

*NASA Jet Propulsion Laboratory (JPL) California Institute of Technology (Caltech)
Structural Cognizant Engineer OPALS Project

ABSTRACT

Computational Fluid Dynamics (CFD) using the commercial CFD package CFX has been performed at NASA Jet Propulsion Laboratory (JPL) California Institute of Technology (Caltech) in support of the Phaeton Early Career Hire Program's Optical Payload for Lasercomm Science (OPALS) mission. The OPALS project is one which involves an International Space Station payload that will be using forced convection cooling in a hermetically sealed enclosure at 1 atm of air to cool "off-the-shelf" vendor electronics. The CFD analysis was used to characterize the thermal and fluid flow environment within a complicated labyrinth of electronics boards, fans, instrumentation, harnessing, ductwork and heat exchanger fins. The paradigm of iteratively using CAD/CAE tools and CFD was followed in order to determine the optimum flow geometry and heat sink configuration to yield operational convective film coefficients and temperature survivability limits for the electronics payload. Results from this current CFD analysis and correlation of the CFD model against thermal test data will be presented. Lessons learned and coupled thermal / flow modeling strategies will be shared in this paper.

INTRODUCTION

This paper documents the recent design and analysis of a Commercial Off the Shelf (COTS) driven space mission which involved the use of forced convection cooling as the heart of its active thermal control system. The Optical Payload for Lasercomm Science (OPALS) project at NASA Jet Propulsion Laboratory (JPL) California Institute of Technology (Caltech) was a new-career hire program wherein the Systems Engineering team and the Cognizant Engineers (CogE) were all within the first few years of their engineering careers. As such, JPL mentors were assigned to monitor the status of the project from PDR to CDR. During this timeframe the Thermal CogE and Structural CogE worked in close synergy with the Computational Fluid

Dynamics (CFD) analyst in order to characterize the fluid flow and heat transfer behavior of the sealed enclosure sub-system.

The CFD task was constrained by the overall thermal requirements placed on the project as well as by time and budget demands. As is typical with CFD analysis, several computer simulations were required in order to optimize the flow geometry. Subsequently several more “production” simulations were needed in order to exercise the model across the trade space of the given problem’s parameter ranges. This paper will focus on the details of the CFD modeling process and document the growth of the analysis as it reached maturity.

The results of CFD provide valuable flow-field visualization feedback on the validity of a proposed flow-tailoring geometry design. In addition, CFD analysis bridges the gap between handbook correlations for the convective film coefficient “h-value”, which can be off by as much as 25% per Incropera and Dewitt ¹, and actual test data. From first-hand experience, the primary author of this paper has witnessed uncertainties in theoretical convective film coefficient values up to 50%, especially when dealing with two-phase flows and/or modeling heat pipe thermo-physics. To this end, CFD is a valuable tool in the prediction of the h-value for convection dominated flow problems. These h-values derived from CFD can then later be used in system’s level models of the overall hardware in order to correlate the thermal model to actual on-station predictions. In closing, CFD can be used to mitigate some of the risks associated with determining the convective film coefficient for a given sub-system, and should only be used as a design guide. In all situations, an engineering unit level test is highly recommended in order to fully characterize and understand the h-values for a given piece of hardware.

DESIGN APPROACH

Figure 1 documents the OPALS mission with respect to its location on the International Space Station (ISS). The payload subsystem configuration is shown in Figure 2. The focus of the present CFD analysis was on the Sealed Enclosure sub-system. This is shown in detail in Figure 3. The interior of the sealed enclosure sub-system’s cylindrical pressure vessel contains the various COTS avionics boards and cooling fans.

Details of the internal parts housed within the pressure vessel are shown below in Figures 4 through 6. The primary active thermal control system is comprised of the following components: heat exchanger, radiator, component tray, duct and cooling fans. The component tray houses the various COTS avionics boards, as well as the power board electronics unit. Vendor supplied COTS cooling fans are selected to meet the thermal design requirements, typically the fans are rated at 120 cfm and have 4 inch diameter blade. The heat exchanger, radiators and duct are custom manufactured items. The heat exchanger utilizes an array of fins in order to enhance the convection heat transfer performance. This fin array has been designed with COTS vendor hardware dimensions in mind.

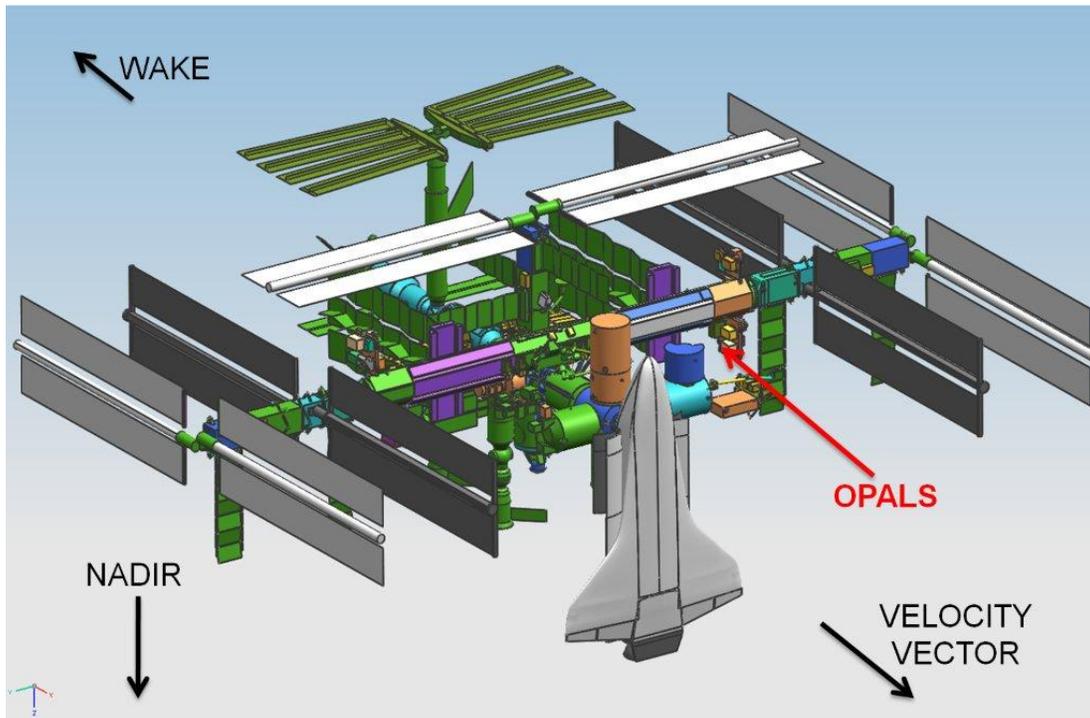


Figure 1. OPALS payload on the ISS.

Figure 2 shows the various sub-systems contained within the OPALS payload.

Flight System

- Optical Head
 - Beacon Acquisition Camera
 - Downlink Transmitter
 - 2-axis Gimbal
- Sealed Electronics Box
 - Laser
 - Avionics
 - Power distribution
 - Digital I/O board

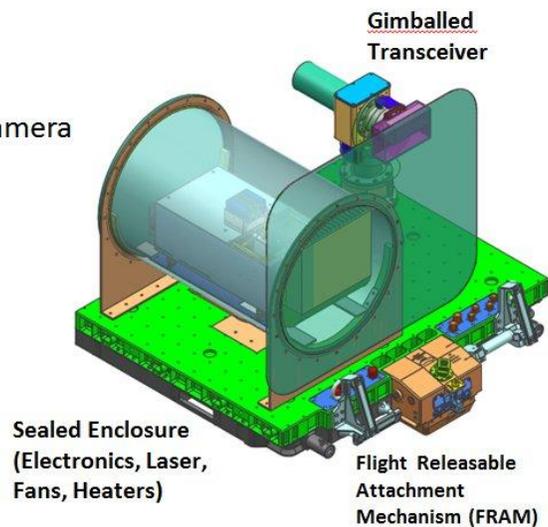


Figure 2. OPALS payload sub-systems.

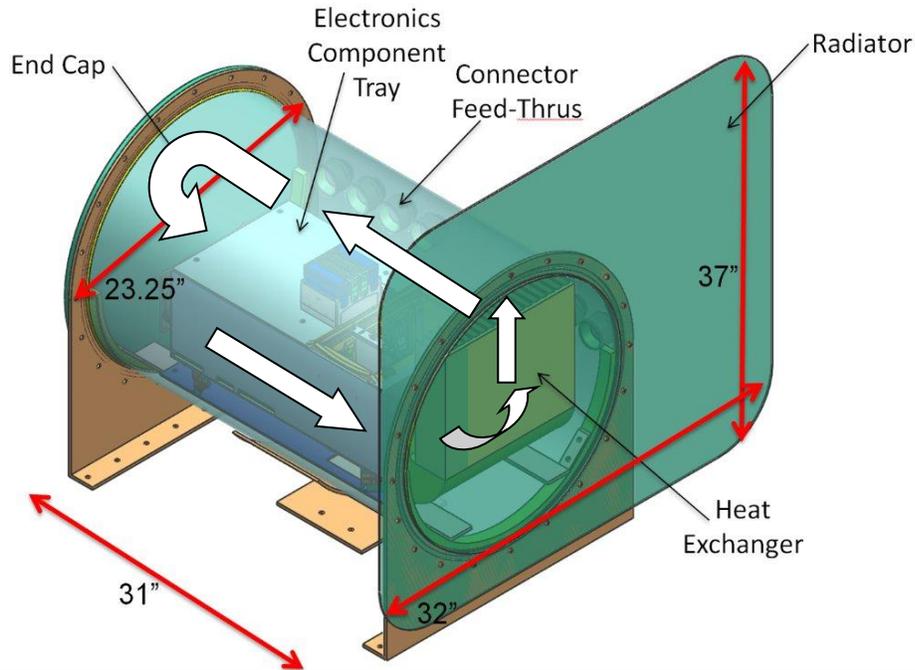


Figure 3. OPALS sealed enclosure sub-system, block arrows indicate flow path.

The overall flow path is as follows: the cool air within the pressure vessel is drawn within the component tray housing via the two fans which act in parallel. The cool inlet air flows over the avionics and power board, picking up the dissipated heat via convection, and then this hot air is passed over the heat exchanger which is tied to the radiator. Since the radiator has a view to cold space, the heat liberated by the heat exchanger is radiated. This process repeats itself as the air circulates from the top exit of the heat exchanger and is drawn to the inlet of the fans.

As can be gathered from Figures 4 through 6, the flow path involved in the thermal control system is rather complicated. This CFD analysis lasted approximately 18 months, a relatively short time frame illustrating how complicated a CFD analysis can be, even for a sub-system as simple as the one considered herein.

During the analysis period of the OPALS project, the design of the thermal control system matured greatly. In order to give the reader the proper perspective, Figure 7 shows the initial design concept when the CFD analyst was first tasked to analyze the flow field and heat transfer of this sub-system.

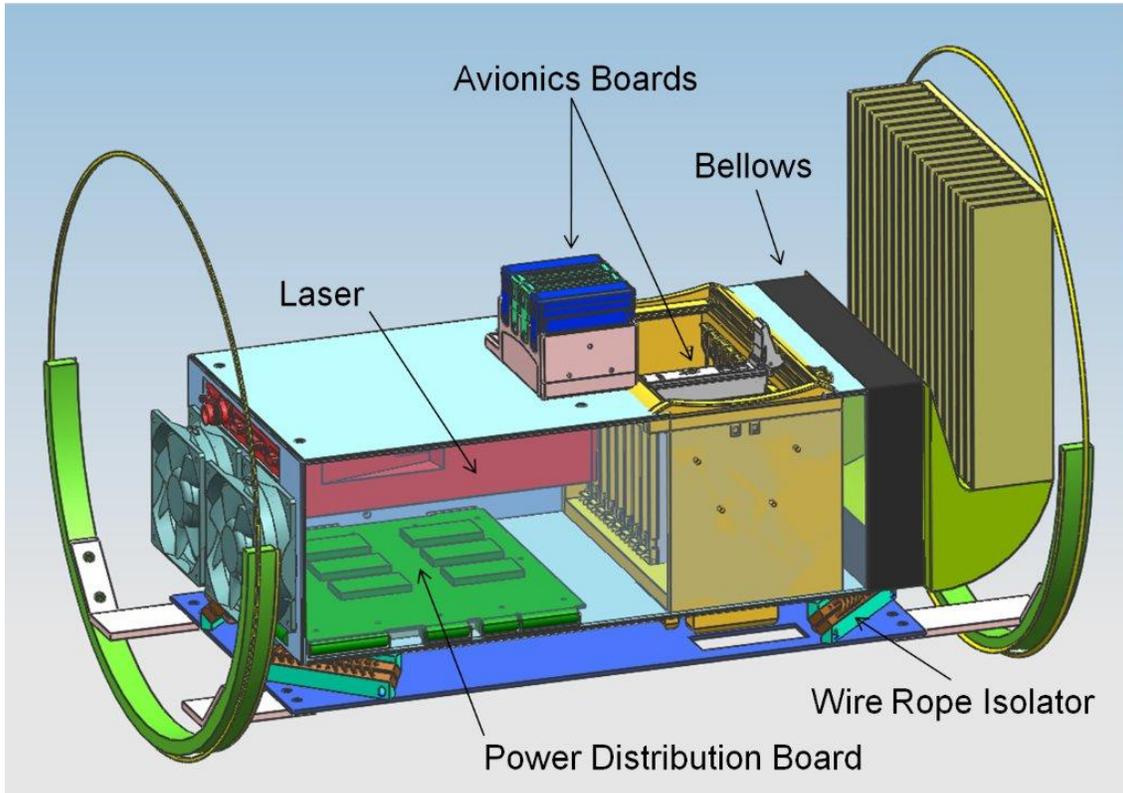


Figure 4. OPALS sealed enclosure pressure vessel internal component.

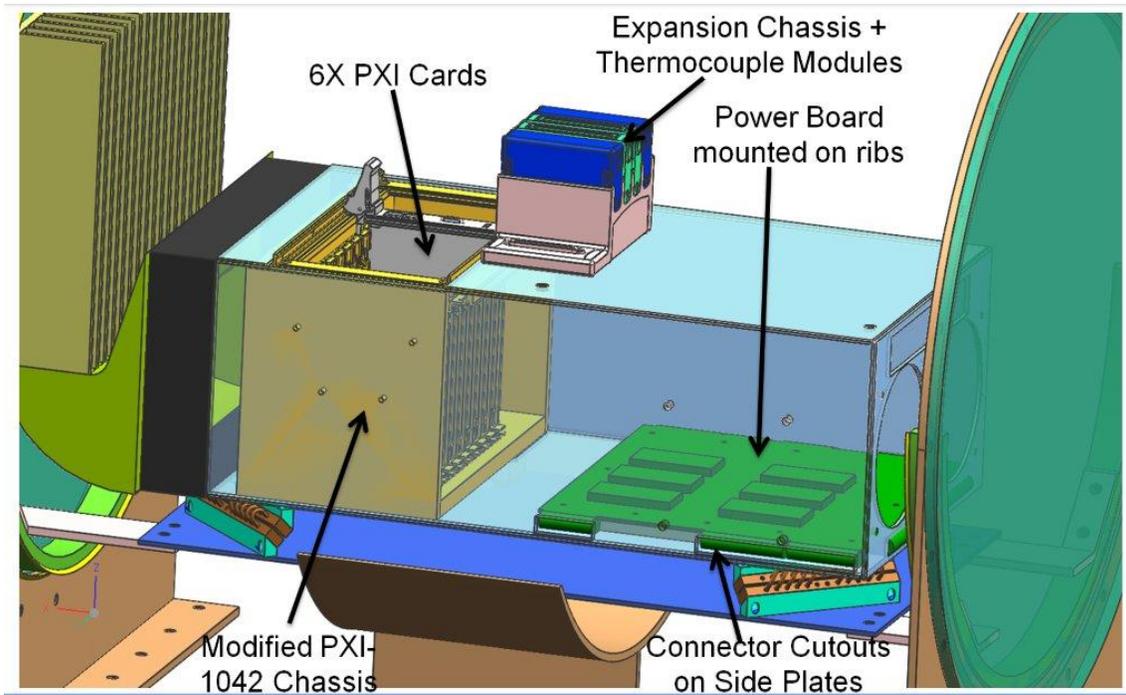


Figure 5. OPALS sealed enclosure pressure vessel internal component.

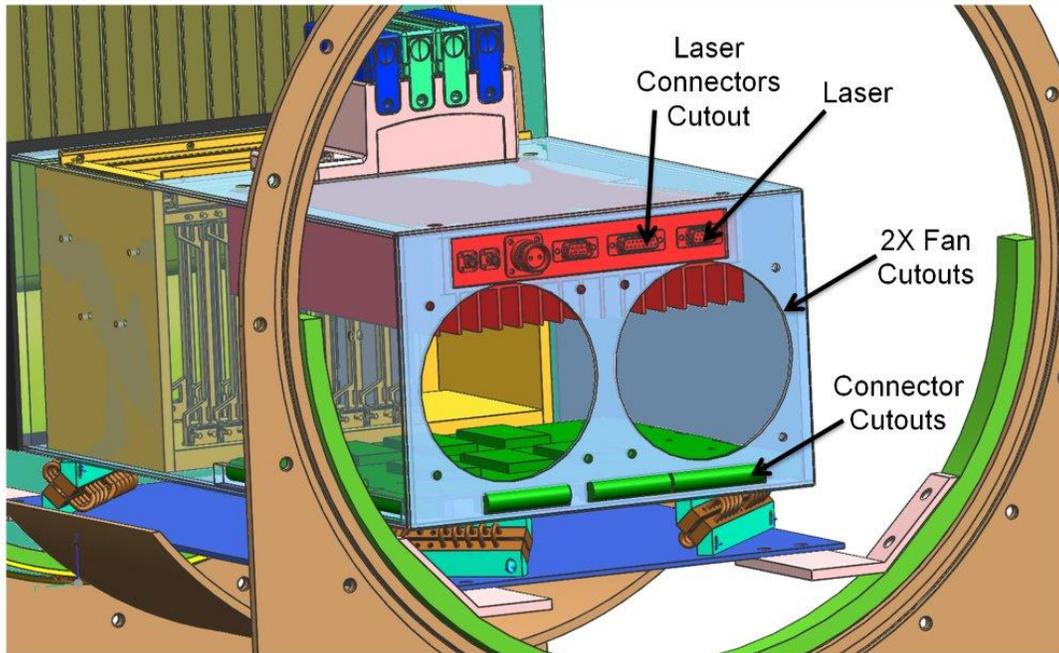


Figure 6. OPALS sealed enclosure pressure vessel internal component.

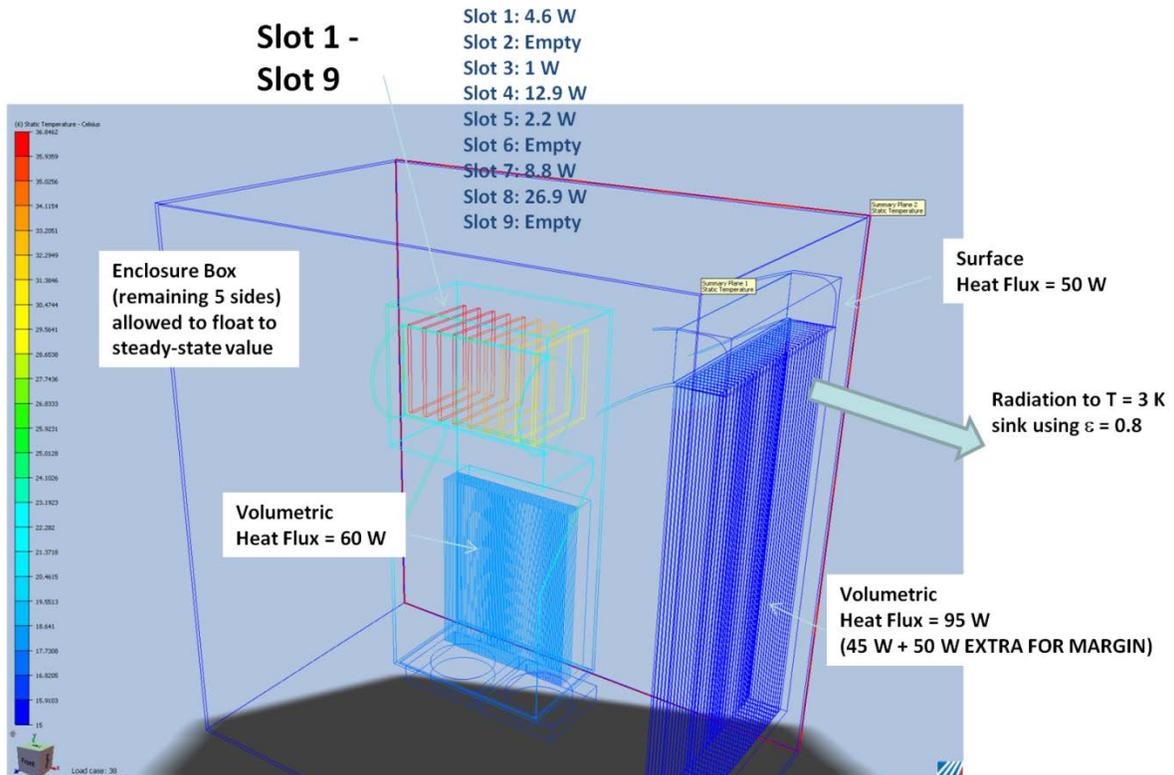


Figure 7. Initial design concept for active thermal control system of sealed enclosure (color bar scale; blue = 15 °C, green = 25 °C, yellow = 30 °C, red = 37 °C).

As can be seen, by comparing Figure 3 (final concept) to Figure 7 (initial concept) several improvements were made in the flow path of the thermal control system. For instance, the final design virtually (with the exception of the interface between the duct and the heat exchanger) eliminates any drastic 90 degree turns in the flow field. More importantly, the final design utilizes a cylindrical pressure vessel, which aids in tailoring the flow field to be more axisymmetric in nature. The initial design concept did not employ the use of a radiator. Using a radiator on the final design allowed for the design of a more compact heat exchanger unit which could be housed within the cylindrical pressure vessel's geometric constraints. It should be mentioned that the Structural CogE was very instrumental in the architecture of the flow network design. Working closely with the Thermal CogE and the CFD analyst, the Structural CogE was able to streamline the flow geometry to the state of its final design. This of course took several iterations.

The thermal design guidelines per Cengel² listed in Table 1 were adhered to when selecting the fans and sizing the flow path associated with electronics cooling.

Table 1. Thermal Guidelines for Selecting Fans for Electronics Cooling per Cengel²

Guideline Number	Guideline
1	CHECK IF NATRUAL CONVECTION IS SUFFICIENT
2	1. SELECT A FAN NEITHER TOO SMALL OR TOO LARGE <ul style="list-style-type: none"> – UNDERSIZED = OVERHEATING – OVERSIZED = MORE EXPENSIVE AND CONSUME MORE POWER
3	MOUNT THE FAN AT THE INLET OF THE BOX AND FILTER THE AIR TO KEEP DIRT OUT
4	POSITION THE EXIT FOR ADEQUATE AIR FLOW THROUGHOUT THE BOX
5	PLACE CRITICAL COMPONENTS NEAR THE ENTRANCE, WHERE THE AIR IS COOLEST
6	ARRANGE PCBs SUCH THAT FLOW RESISTANCE IN THE BOX IS MINIMIZED
7	CONSIDER THE EFFECTS OF ALTITUDE, h-VALUES SCALE WITH SQUARE ROOT OF LOCAL PRESSURE
8	ARRANGE THE SYSTEM SO THAT NATURAL CONVECTION AIDS THE FORCED CONVECTION, I.E. MOUNT THE PCBs VERTICALLY AND BLOW THE AIR FROM BOTTOM TO TOP

Table 1 (continued). Thermal Guidelines for Selecting Fans for Electronics Cooling per Cengel²

Guideline Number	Guideline
9	<p>AVOID FLOW SECTIONS WHICH INCREASE THE FLOW RESISTANCE</p> <ul style="list-style-type: none"> – I.E. UNNECESSARY CORNERS, SHARP TURNS, SUDDEN EXPANSIONS, AND/OR CONTRACTIONS, AND VERY HIGH VELOCITIES ($> 7 \text{ m/s}$) SINCE THE FLOW RESISTANCE IS PROPORTIONAL TO THE FLOW RATE ($h \propto Q^2$) – AVOID VERY LOW VELOCITIES, SINCE THEY RESULT IN POOR HEAT TRANSFER AND ALLOW DUST/DIRT IN THE AIR TO SETTLE ON THE COMPONENTS
10	<p>FOR USE OF TWO OR MORE FANS, DECIDE WHETHER TO MOUNT THE FANS IN SERIES OR IN PARALLEL</p> <ul style="list-style-type: none"> – FANS IN SERIES WILL BOOST THE PRESSURE HEAD, THUS BEST FOR SYSTEMS WITH A HIGH FLOW RESISTANCE – FANS IN PARALLEL WILL INCREASE THE FLOW RATE, THUS BEST FOR SYSTEMS WITH SMALL FLOW RESISTANCE
11	<p>USE CFD TO EXERCISE 1. THROUGH 10. ABOVE (THIS GUIDELINE HAS BEEN ADDED BY THE PRIMARY AUTHOR OF THE PRESENT PAPER)</p>
12	<p>PERFORM AN ENGINEERING UNIT LEVEL TEST (THIS GUIDELINE HAS BEEN ADDED BY THE PRIMARY AUTHOR OF THE PRESENT PAPER)</p>

Following the suggestions listed in Table 1 led to the improvements realized in the final design revision. This necessarily involved several iterations between the Structural CogE, Thermal CogE, and the CFD analyst, resulting in numerous time-consuming (CPU intensive) 3-D simulations using the CFD software. In order to aid in the turn-around process, the turn-key CFD tool CFDesign 2012 was chosen to analyze the flow/heat characteristics of this sub-system. At that time CFDesign 2012 was the default CFD software available at NASA/JPL. The CFDesign tool is a very user-friendly simulation tool and as such having a small learning curve allows analysis engineers to come up to speed quickly on how to use the software. Next, the methodology used to create the CFD model is discussed.

CFD Modeling Methodology

This CFD code used was Blue Ridge Numeric's CFDesign 2012. This code is a finite element based CFD heat/flow solver. The equations of motion being solved are the full Navier-Stokes with Conservation of Energy, Conjugate Heat Transfer, and the $k-\epsilon$ turbulent closure model. The CFDesign code using Galerkin FEM with pressure correction to formulate and solve the equations of motion. The modeling flow path is as follows:

1. Read in CAD geometry into CFDesign as Parasolid file
2. Create material data blocks
 - a. CFDesign has a FAN object included in its material's database, whereby various vendor supplied fan curves can be digitized and implemented in the flow solution to match flow (cfm) vs. head (in W.G.) and optimize the problem
 - b. The FAN can also be tied to a THERMOSTAT controller in CFDesign, enabling the user to simulate ON/OFF control of a region of flow with FAN
3. Create B.C.'s / I.C.'s
 - a. Dirichlet ($T = \text{const.}$, $Q = \text{const.}$)
 - b. Von Nuemann
 - c. h-value to a fluid sink temperature
 - d. Stefan-Boltzmann radiation to a sink
4. Mesh
 - a. CFDesign uses an auto-mesher and TETRAHEDRAL elements
 - b. Limited, but robust, able to accurately resolve turbulent boundary layers
5. Solve x, y, z momentum equations
6. Solve pressure correction equation
7. Correct velocities via pressure correction based on the Patankar³ SIMPLE algorithm
8. Solve energy equation
9. Solve Turbulent Kinetic Energy equation
10. Solve Turbulent Kinetic Energy dissipation equation
11. Check convergence (goto 5)
12. Perform output calculations
13. Write out data
14. Perform post-processing, i.e. plot h-values, etc.

Design Cases Simulated

Various scenarios were simulated in this investigation. The primary driving thermal requirement which was levied by the COTS National Instruments Data Boards, was that the air temperature of the air near the inlet to the card array could not exceed 50 °C. Other pertinent design assumptions were:

- ISS-provided thermal environment definition
- Stacked worst-case assumptions serve to bound environmental loads
 - Seasonal and EOL/BOL variations
 - Maximally constrained radiation windows
 - Adiabatic interface assumed between payload and ISS.
- With regards to the flow design the following methodology was adopted:
 - Minimalist approach for convective cooling system – keep it simple
 - Single flow path with major heat sources in series
 - Dry air selected on basis of cost and practicality with only modest sacrifices in performance
 - Eliminates additional purge/fill valves
 - Easier to test-as-you-fly
 - Insulated interface between electronics and structure
 - Single heat exchanger, strongly-coupled to radiator
 - National Instruments cPCI hardware, designed for use in a room temperature environment with fan cooling
 - Thermal control approach aims to provide a forced-convection cooled environment matching as closely as possible that intended by the vendor
 - Laser and Power Board mimic heat dissipation of NI power supply unit
 - Component Tray is thermally isolated from enclosure structure by design
 - Fan performance is primary source of margin

- Stock NI cooling fans provide ~120cfm*
- Replacement COTS fans could provide up to 180cfm*
- MIL-spec option could provide up to ~400cfm*

* = Absent pressure losses due to friction

The design cases listed in Table 2 were analyzed over the course of the project as identified by the Thermal CogE.

Table 2. Design Cases Simulated in this CFD Study

Design Case	Description
1	Worst case hot steady state
2	Thermal Engineering Unit Thermal Test Data Simulation / Model Correlation
3	On-station flow blockage / harness / strap simulation
4	Survival heater sizing design study
5	Systems level SINDA model h-value correlation using CFD results

Each of these is discussed in detail below.

Worst case hot steady state

Figures 8 through 11 show the set-up for a typical worst case hot analysis. Typical run-time parameters and inputs into the model were as follows: solid elements~ 300,000, fluid elements ~1,500,000, steady state run-time to convergence (300 iterations) was 6 hrs. on a 64-BIT dual core Windows workstation. The following boundary conditions were used: radiation sink from radiator to 3 K using $\epsilon=0.8$. Environmental flux of 239 W/m² applied to radiator, heat loads were as follows: cards = ~ 60 W, laser = 60 W, power board= 44 W. Initial conditions of the air were 10 °C, 14.7 psia. The fan speed was 60 CFM per fan (there were 2 fans used, the nominal 120 CFM per fan was de-rated to 60 CFM per fan to account for friction and to add margin to the design). From post-processing a fan speed of 90 CFM (180 CFM total) yields an maximum film coefficient value of $h = 30 \text{ W/m}^2\text{-K}$ on the heat exchanger. This value agreed with the Thermal Engineering Unit Thermal Test Data Simulation / Model Correlation as discussed later in this paper.

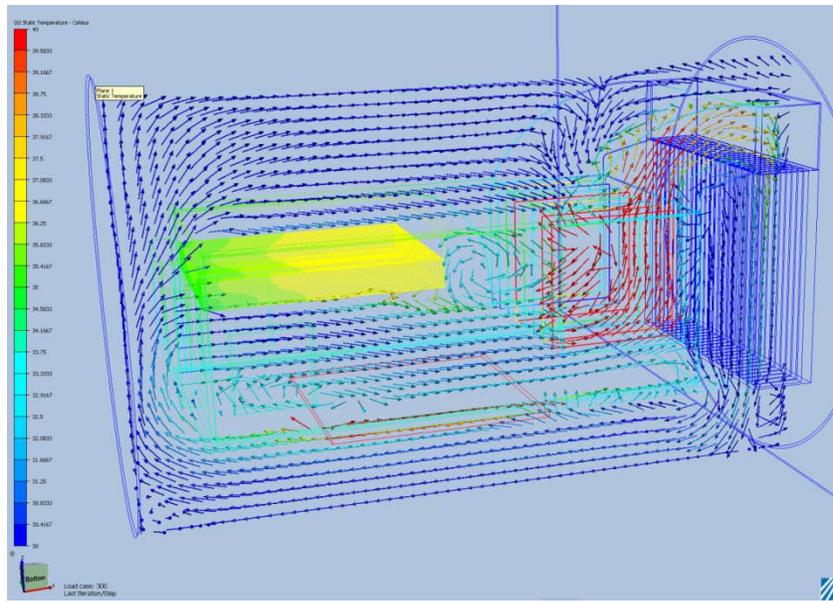


Figure 8. Worst case hot CFD simulation velocity vectors colored by air temperature (color bar scale; blue = 30 °C, green = 35 °C, yellow = 37 °C, red = 40 °C).

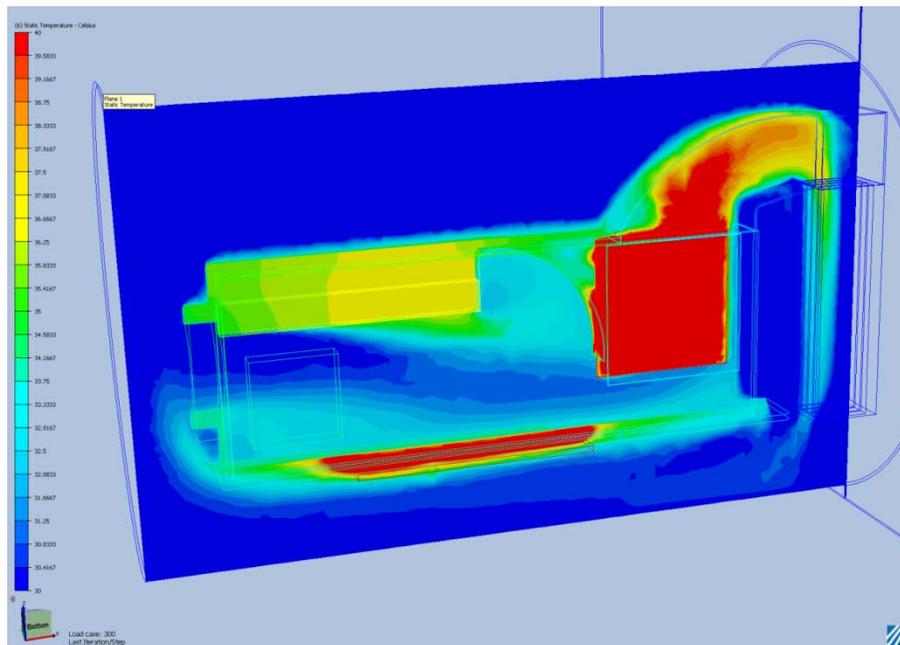


Figure 9. Worst case hot CFD simulation air temperatures (color bar scale; blue = 30 °C, aquamarine = 33 °C, green = 35 °C, yellow = 37 °C, red = 40 °C).

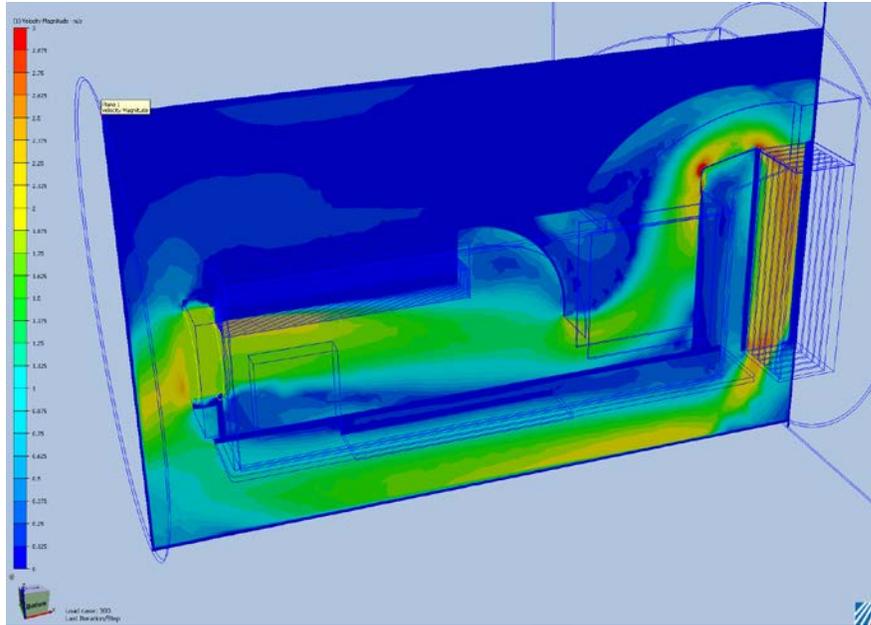


Figure 10. Worst case hot CFD simulation velocity magnitude (color bar scale; blue = 0 m/s, green = 1.5 m/s, yellow = 2 m/s, red = 3 m/s).

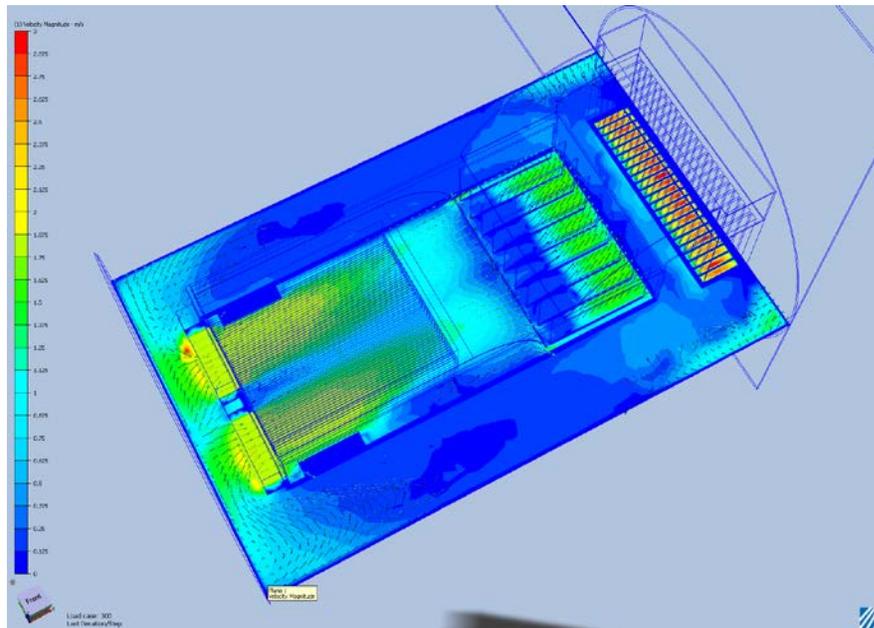


Figure 11. Worst case hot CFD simulation velocity vectors colored by velocity magnitude (color bar scale; blue = 0 m/s, green = 1.5 m/s, yellow = 2 m/s, red = 3 m/s).

The results of the worst-case hot simulation show that the flow control system is working as desired. From Figure 9, we see that the hottest air temperature is 40 °C near the card inlets. This satisfies the primary design requirement of having the air at the card inlet less than 50 °C. The flow path is illustrated in Figures 10 and 11. Figure 10 shows that the primary flow is confined to the “fan/duct/heat exchanger” network as engineered. The velocity vectors shown in Figure 10 indicate the draw-through which the fans entrain the surrounding fluid in the sealed enclosure. This surrounding fluid is at a lower temperature than that of the air exiting the heat exchanger, thus as this colder air is drawn into the duct the cooling system can perform the heat removal process again and repeat the desired cycle.

Thermal Engineering Unit Thermal Test Data Simulation / Model Correlation

The engineering test unit CFD model was used to correlate the thermal test setup CFD model. The geometry of the engineering test unit employed an extra heat exchanger mounted on the back of the internal heat exchanger in lieu of the radiator panel. This geometry was modeled in the CFD software. Figures 12 through 13 show the results of the engineering test unit CFD analysis. Figure 14 shows the test data gathered via thermocouple instrumentation.

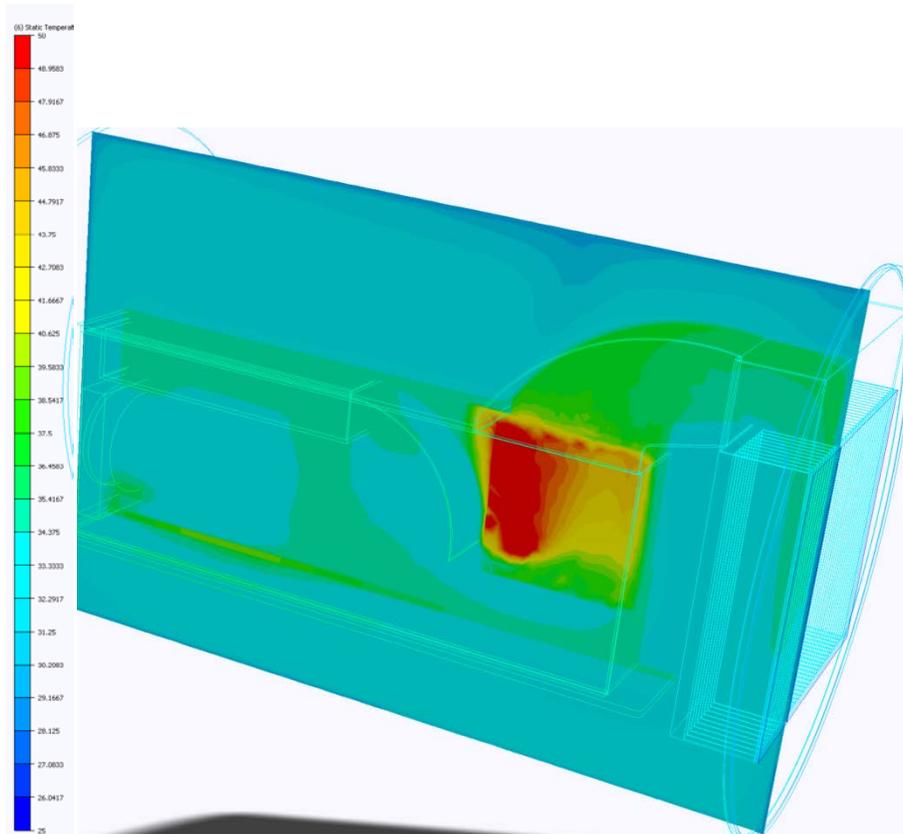


Figure 12. Engineering test unit CFD simulation enclosure air temperatures (color bar scale; blue = 25 °C, aqua-marine = 33 °C, green = 38 °C, yellow = 42 °C, red = 50 °C).

From Figure 13, it is clear that the engineering test unit under operation meets the primary thermal design requirement of having the air at the inlet of the card array be less than or equal to 50 °C. In fact, the majority of the air within the sealed enclosure's cylinder is running at a value of 34 °C. The hot spot of 50 °C is in the proximity of the controller card as expected.

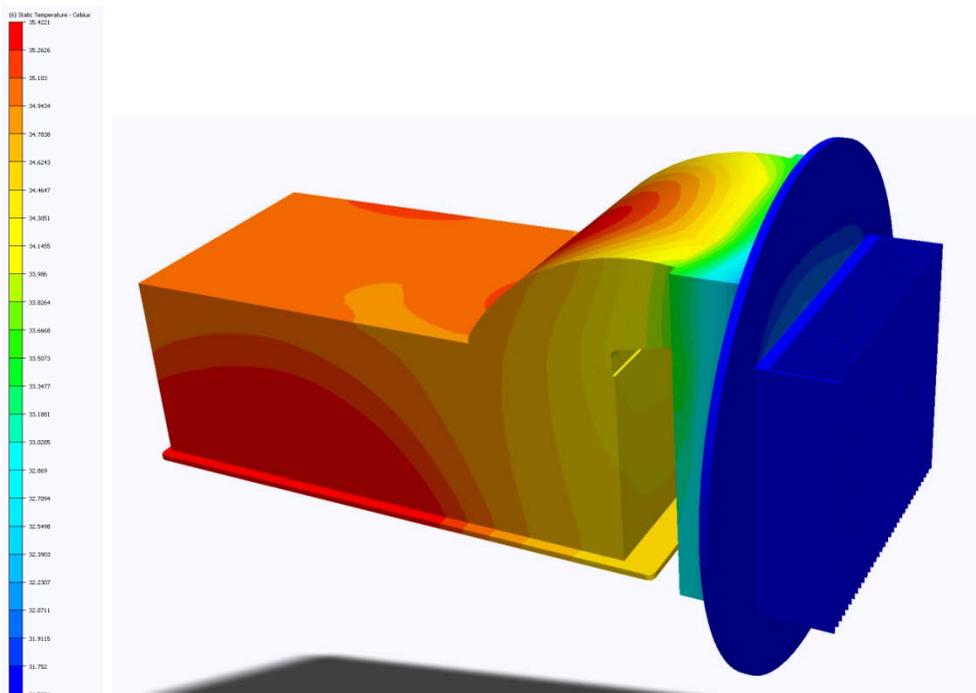


Figure 13. Engineering test unit CFD simulation component tray temperatures (color bar scale; blue = 32 °C, green = 33 °C, yellow = 34 °C, red = 35 °C).

Figure 13 illustrates that the CFD simulation of the engineering test unit predicts the component shelf (which houses the cooling fans, COTS avionics boards, etc.) of the sealed enclosure to run at a minimum of 31 °C and a maximum of 36 °C. The thermocouple telemetry data of Figure 14 shows that, under test, the internal components range in temperature from 30°C to 35 °C. Thus, comparing Figure 13 to Figure 14, we see that the thermal model matches the test data to within 2 °C.

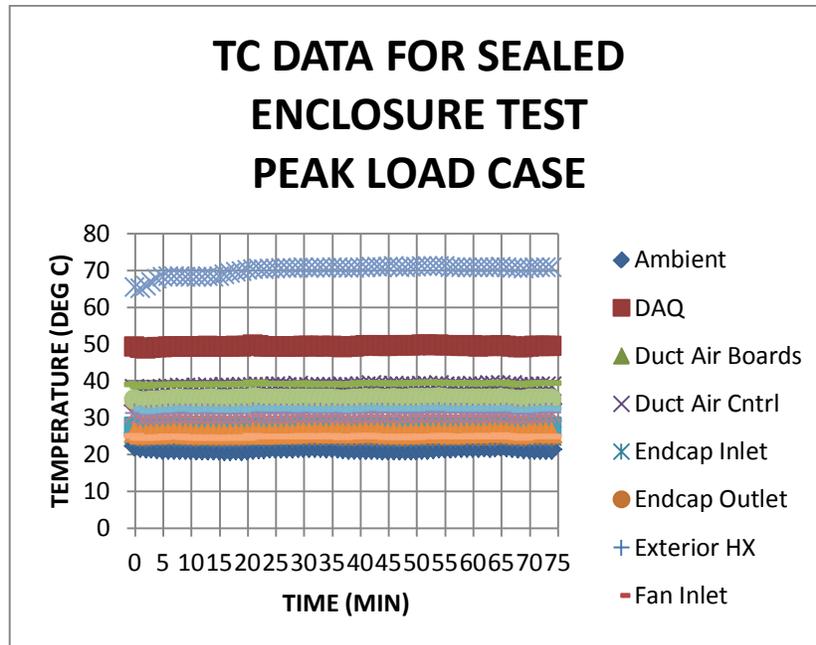
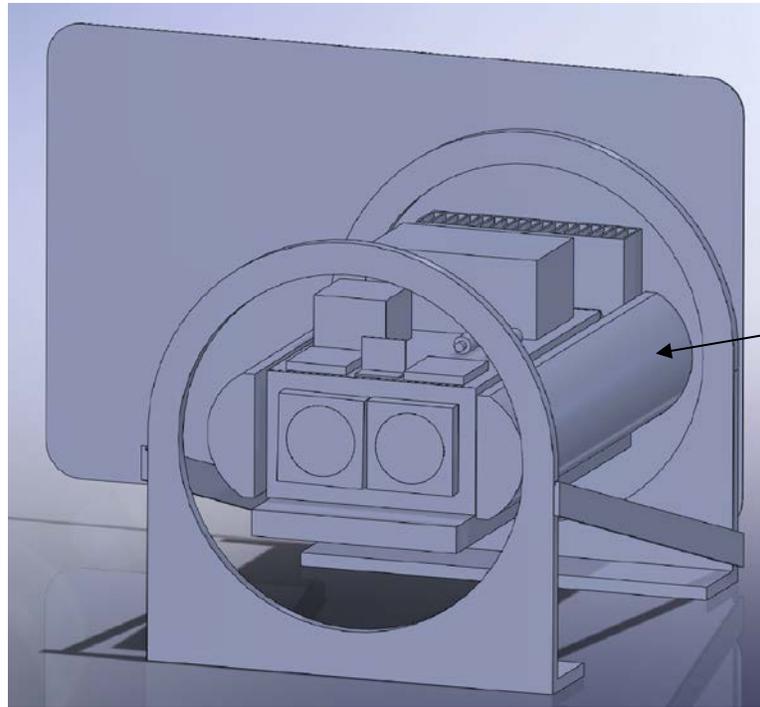


Figure 14. Engineering test unit thermocouple telemetry data.

On-station flow blockage / harness / strap simulation

Upon final assembly, the sealed enclosure incorporates several feed-throughs, and wire harnessing within the sealed enclosure cylinder. This additional wiring and harnessing act as a flow impedance. In order to mitigate risk, a final CFD model was built, which served to mimic the effects of this additional flow impedance. Here a flowrate of 90 cfm was used for each fan. This was used to give the nominal value of 60 cfm extra margin to overcome the pressure drop developed by the flow blockages. In addition, the diagonal mounting straps were modeled in order to ascertain the thermal gradient set-up within them. The flow blockage due to harnessing, feed-throughs, and telemetry cabling was simulated in CFD by placing large flow blockage regions within the sealed enclosure. Figures 15 through 20 show the results of this flow blockage simulation. Figure 15 shows the CAD model used to add in flow blockages, and also the diagonal straps.



Flow blockage geometry added to mimic harness, etc.

Figure 15. Flow blockage CFD simulation geometry set-up.

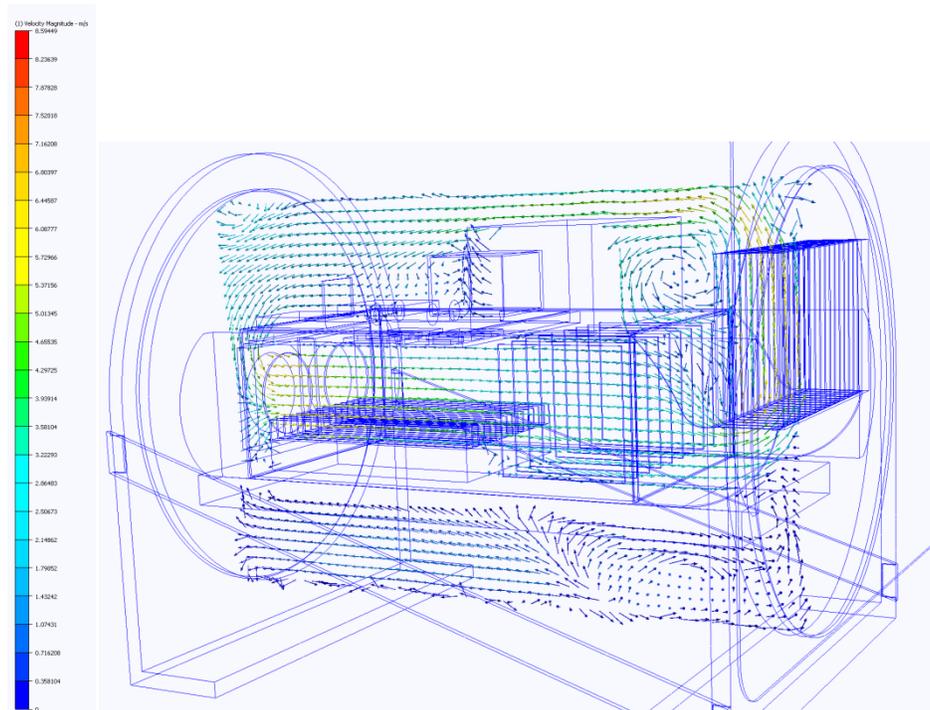


Figure 16. Flow blockage CFD simulation velocity vectors colored by velocity magnitude (color bar scale; blue = 0 m/s, green = 4.0 m/s, yellow = 6.5 m/s, red = 8.6 m/s).

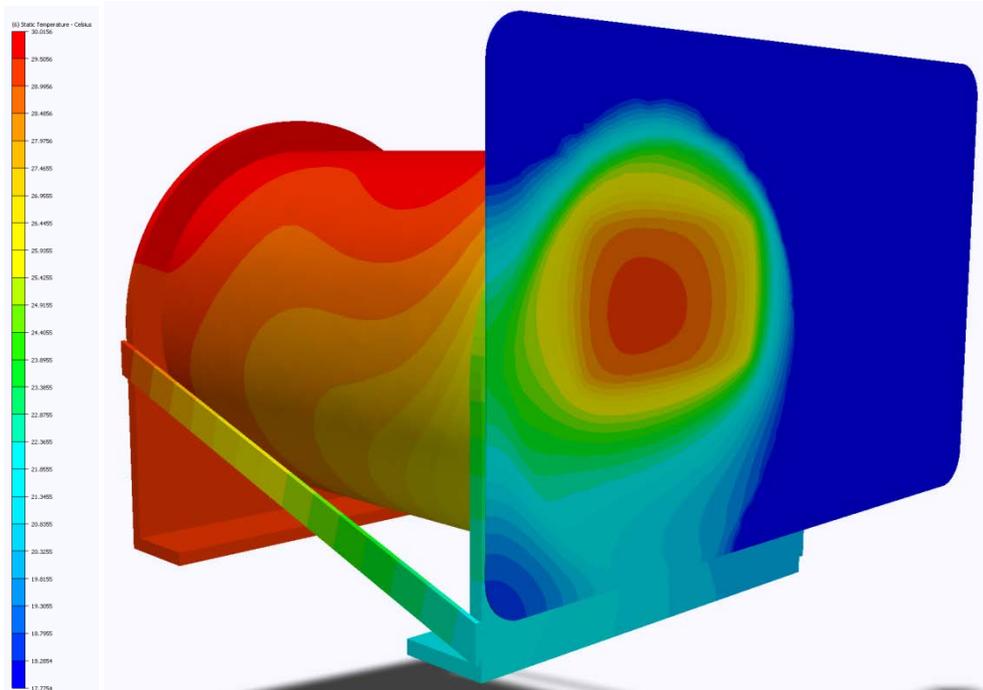


Figure 17. Flow blockage CFD cylinder, radiator and strap isotherms (color bar scale; blue = 18 °C, green = 24 °C, yellow = 27 °C, red = 30 °C).

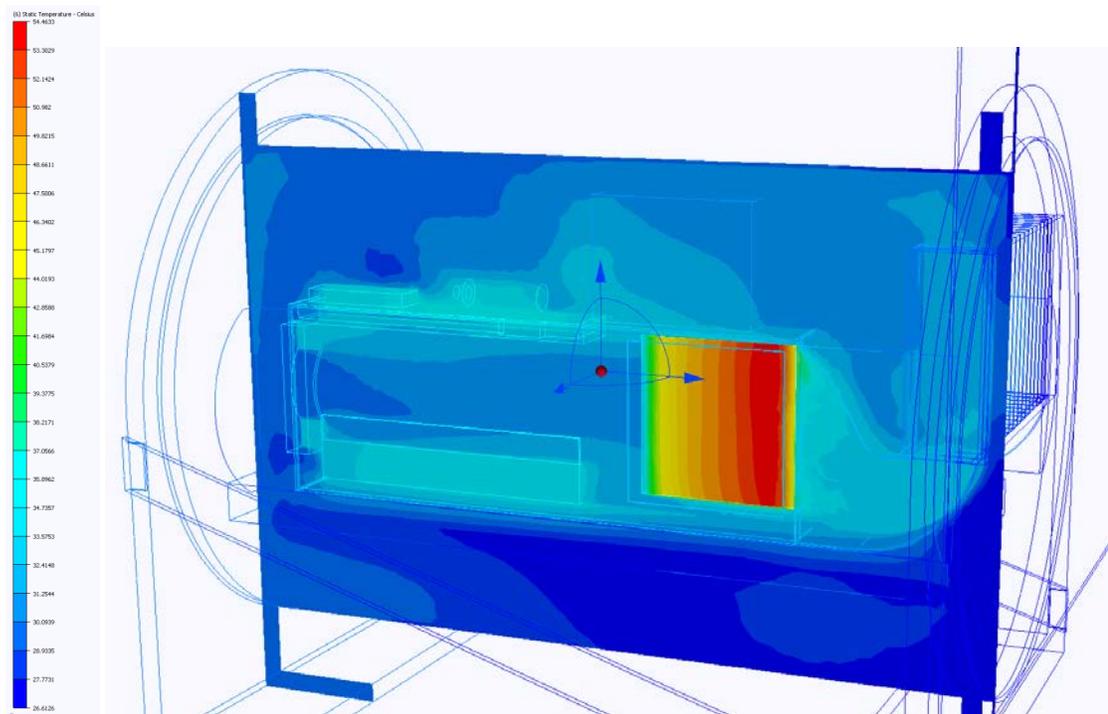


Figure 18. Flow blockage CFD sealed enclosure air temperature in proximity of COTS NI cards (color bar scale; blue = 27 °C, green = 42 °C, yellow = 48 °C, red = 54 °C).

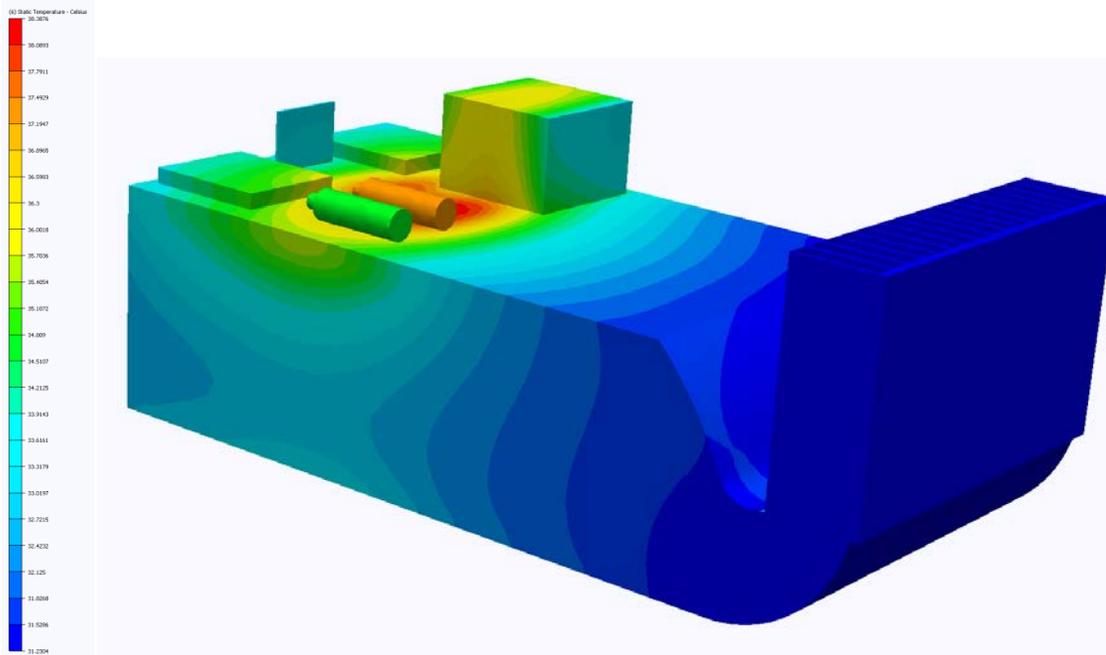


Figure 19. Flow blockage CFD component tray isotherms (color bar scale; blue = 31 °C, green = 35 °C, yellow = 37 °C, red = 38 °C).

Figure 16 shows the velocity vectors for the flow-blockage simulation. With the addition of the added flow impedances, the primary flow path still remains to be the circuit of “fans/duct/heat exchanger”, as desired. Figure 16 shows the region or re-circulation in the vicinity of the heat exchanger and the thermocouple block on the top of the component tray. From the velocity vectors in Figure 16, the inlet to the fans corresponds to average velocity 5.2 m/s, which for a 4 inch diameter fan leads to a average volumetric flow rate of 90 cfm

Figure 17 shows the effect of the radiator, i.e. there is a cold spot on the upper right hand corner of the radiator, as expected. The local hot spot between the radiator and the heat exchanger interface has been captured in this conjugate heat transfer coupled fluid flow simulation as well. By prescribing an interfacial contact conductance between the radiator and the heat exchanger of $h = 1500 \text{ W/m}^2\text{-K}$, we have simulated the presence of a bolted thermal gasket between the cylinder end cap and the radiator. As seen in Figure 17, there is a delta of about 5 °C between the center of the heat exchanger and the radiator. This is in agreement with typical temperature delta’s witnessed across gasketed/bolted thermal interfaces. Also, from Figure 17 the gradient along the external strap is found to be approximately 6 °C.

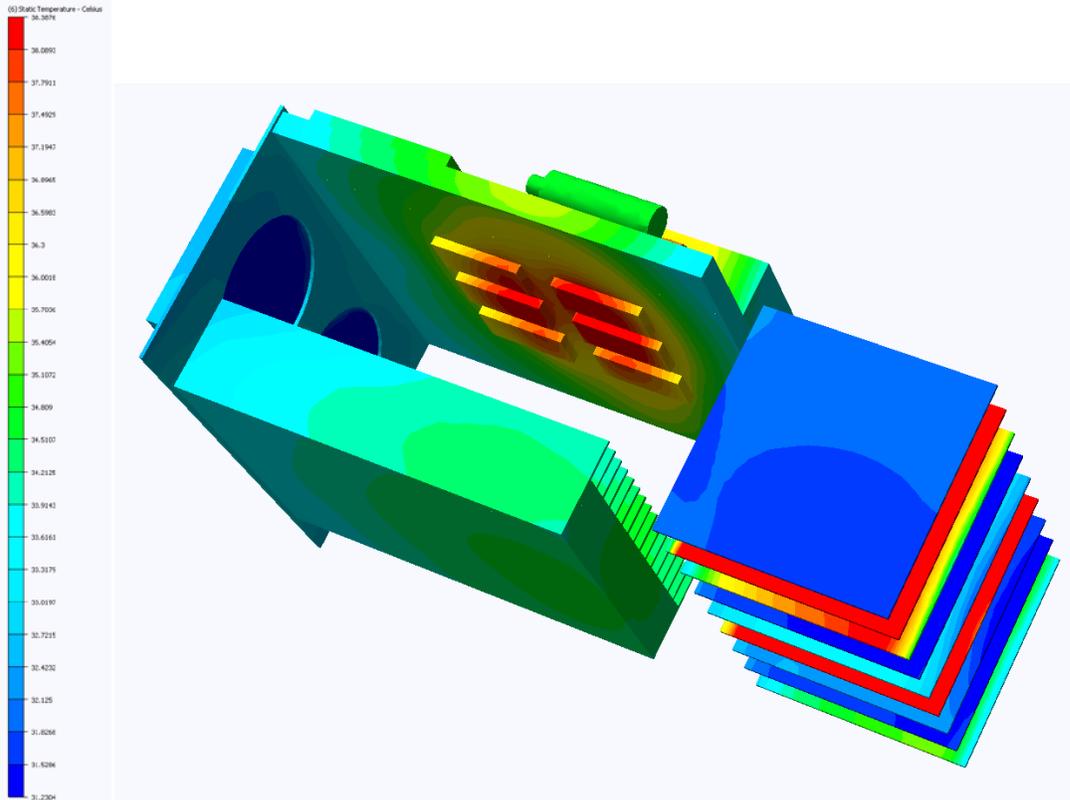
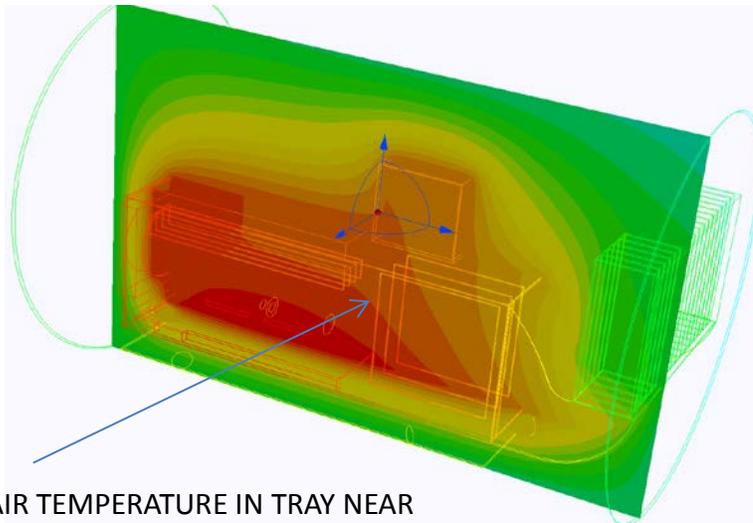


Figure 20. Flow blockage CFD laser, power board, control card isotherms (color bar scale; blue = 31 °C, green = 35 °C, yellow = 36 °C, red = 38 °C).

Figure 18 shows again that, even in the presence of excessive flow blockage that the thermal design requirement of having the air at the inlet of the cards less than 50 °C is satisfied. Figure 18 shows the isotherms of the air in the sealed enclosure at a vertical cross-section in the vicinity of the controller card inlet. The isotherms show that the inlet air temperature of the cards is approximately 38 °C, well within the requirement of 50 °C. Finally, Figures 19 and 20 indicate that the components within the sealed enclosure are within acceptable temperature limits (with the component hardware having an average of 35 °C).

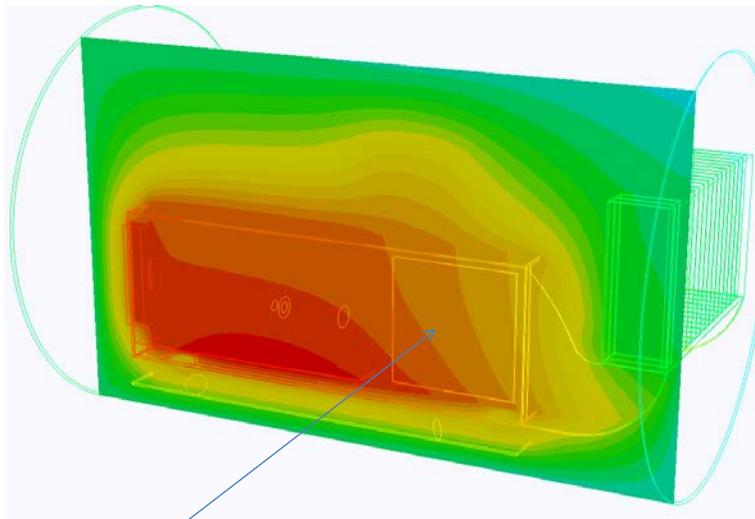
Survival heater sizing design study

This series of CFD simulations were performed in order to assess the correct heater power rating for the survival heaters. Figures 21 through 23 show the results of the survival heater sizing. The following parameters were used for these simulations. All electronics were turned off, including the fans. The air medium was modeled with gas conduction only, i.e. no convection. The radiator was tied to a 3 K sink and had a backload of 33 W. Static heater loads were applied to the heat exchanger back cover (the surface residing within the sealed enclosure and the bottom of the component tray).



AVG. AIR TEMPERATURE IN TRAY NEAR
THE CARDS = 2.7 DEG C

Figure 21. Heater sizing CFD gas-conduction simulation, 30 W applied to back of heat exchanger, 30 W applied to bottom tray surface (color bar scale; blue = -30 °C, green = 30 °C, yellow = 73 °C, red = 115 °C).



AVG. AIR TEMPERATURE IN TRAY
NEAR THE CARDS = 96 DEG C

Figure 22. Heater sizing CFD gas-conduction simulation, 75 W applied to back of heat exchanger, 75 W applied to bottom tray surface (color bar scale; blue = -30 °C, green = 30 °C, yellow = 73 °C, red = 115 °C).

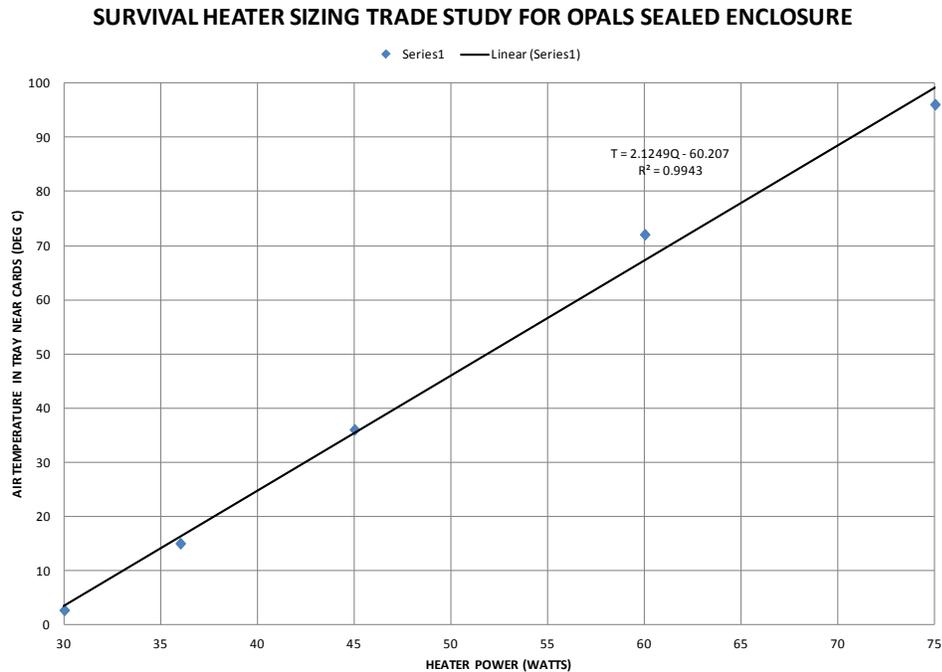


Figure 23. Survival heater sizing CFD trade study.

As can be seen from Figure 21 and Figure 22, the region of the air near the entrance to the controller cards varies quite significantly with the amount of heater power supplied by the survival heaters. To this end a trade study was performed, varying the level of heater power and monitoring the air temperature in the vicinity of the card inlets. The results of this trade study appear as Figure 23. Figure 23 guided the proper selection of the survival heaters, as well as the design of the thermostat algorithm for control of these heaters. The challenge posed in the CFD modeling was the fact that the gas conduction limit in the air within the cylindrical enclosure needed to be modeled correctly. When the convection was disabled, the CFD model reduced to one of a gas-conduction mode. This required a fine mesh size in order for the model to converge correctly, since there no longer any flow velocities.

SINDA model h-value correlation using CFD results

One very useful application of CFD data, is to take the h-values found from the simulations and use them in a systems level SINDA lumped capacitance type of network model. This was the final task of the present CFD analysis. Here, values of the CFD film coefficient found in the simulations were used to correlate a Thermal Desktop (SINDA) node/conductor model of the OPALS sealed enclosure as it interfaces with the Space-X Dragon launch vehicle, which is slated to transfer the OPALS payload to the ISS.

Figures 24 through 25 show the Thermal Desktop node/conductor model of the sealed enclosure. Figure 30 shows the cylindrical sealed enclosure in proximity to the other OPALS hardware. Using the h-values found from CFD, the internal “air nodes” within the Thermal

Desktop model were correlated by adjusting the $G = hA$ value of the SINDA conductor connecting the SINDA air nodes to the cylinder nodes, heat exchanger and component tray.

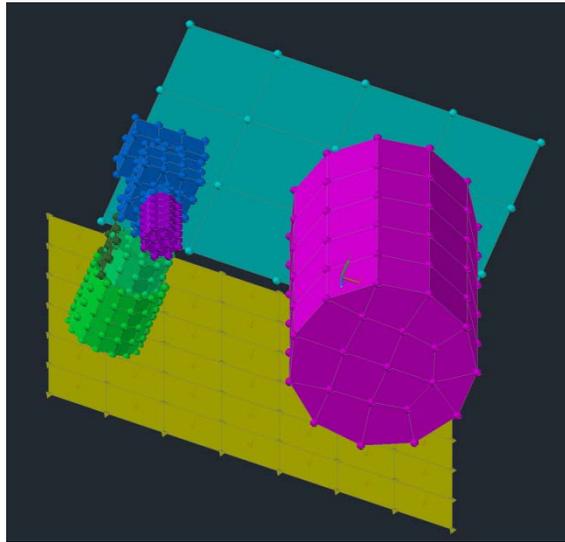


Figure 24. Thermal Desktop systems level thermal model node/conductor model of sealed enclosure.

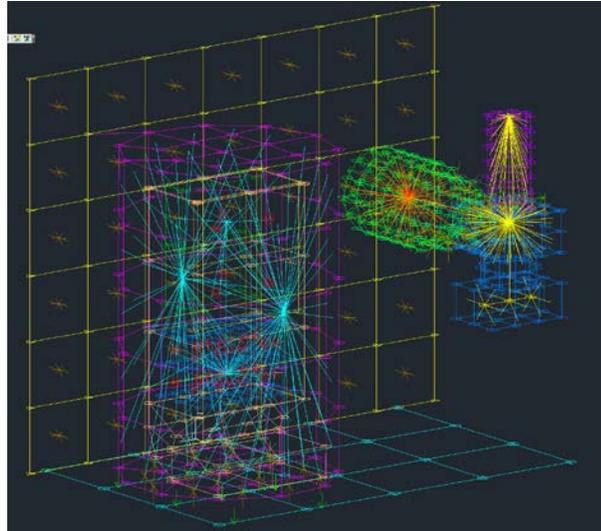


Figure 25. Thermal Desktop systems level thermal model node/conductor model of sealed enclosure (air nodes within the sealed enclosure shown tied to cyan conductors).

To this end, the system's level model will be easier to correlate to actual flight data, since the internal convective environment of the sealed enclosure is already in agreement with the CFD

predicts. As illustrated in Figure 25, the internal “air nodes” are connected by $G=h*A$ convection conductors, which tie the wall of the cylindrical sealed enclosure to the component tray, end caps, heat exchanger and duct, etc. Upon correlation typical values of h within the sealed enclosure ranged from a minimum of $7.5 \text{ W/m}^2\text{-K}$ (in the stagnation regions of the internal air flow) to an average value of $15 \text{ W/m}^2\text{-K}$ (over the area of the heat exchanger). These values are in agreement with forced convection, when comparing to the standard handbook value of natural convection being taken as $h = 5 \text{ W/m}^2\text{-K}$.

CONCLUSIONS

Complicated CFD analysis has been facilitated with a turn-key CFD software package using a low cost, high-end workstation. Several iterations on flow geometry facilitated many highly intensive CFD simulations. The supporting CogE’s worked in close unison with the CFD analyst in order to obtain reliable, accurate results in a short timeframe. Several simulations witnessed during the evolution and CFD analysis of a forced “convective driven” active thermal control system design have been summarized. These included:

1. Worst case hot steady state
2. Transient cool-down
3. Thermal Engineering Unit Thermal Test Data Simulation / Model Correlation
4. On-station flow blockage / harness / strap simulation
5. Survival heater sizing design study
6. Thermal Desktop systems level SINDA model h -value correlation using CFD results

The above demonstrate the versatile nature of scenarios to which CFD can be applied. Obviously, turn-key CFD should be thought of as a design roadmap, and not a final solution. Testing should ultimately be carried out whenever one encounters forced convection comprising the core of the thermal control system. Due to their non-deterministic nature, theoretically derived forced convective turbulent flow heat transfer coefficients can have uncertainties from 25% to 50%. The use of CFD can mitigate the uncertainties involved in predicting/modeling convective flows. The category of desktop based, fast turn-around type of CFD analyses presented herein are necessary to clearly understand any design hurdles one may encounter along the path to development of a project of this caliber. This paper has shown how the close synergy between the CogE’s and the thermal analyst results in a productive work flow path in the development of a forced-convection based thermal control system using COTS hardware.

ACKNOWLEDGEMENTS

The primary author would like to thank Tim George (previously of Blue Ridge Numerics, now with STAR-CCM) and his team for their on-line support. The lead author would also like to thank his co-authors for their time and patience while working on the duration of the OPALS project. Finally, the primary author would also like to thank Eric Sunada for his assistance with the Thermal Desktop portion of this paper. The research was carried out at the Jet Propulsion Laboratory, California Institute of Technology, under a contract with the National Aeronautics and Space Administration. © 2012 California Institute of Technology Government sponsorship acknowledged.

CONTACT

Dr. Kevin R. Anderson, P.E. is a Faculty Part Time Sr. Thermal Engineer for NASA JPL's Thermal and Fluids Analysis Group. Dr. Anderson is also a full-time tenured, professor of Mechanical Engineering at California State Polytechnic University at Pomona (Cal Poly Pomona). Dr. Anderson received his BSME at Cal Poly Pomona in 1991 and his MSME at CU Boulder, 1994 and his Ph.D. in ME at CU Boulder in 1998. Dr. Anderson possesses over 15 years of practical experience working in the aerospace engineering thermal sector. Dr. Anderson has worked at Technology Applications, Inc. in Boulder, CO, Hughes Space & Communications El Segundo, CA, Boeing Satellite Division, El Segundo, CA, Swales Aerospace, Pasadena, CA, ATK Aerospace, Pasadena, CA. Presently Dr. Anderson works at the California Institute of Technology's NASA-Jet Propulsion Laboratory. Dr. Anderson has used CFD packages since 1992, including FLUENT, STAR-CCM, CFD2000, CFX-Design 2012, IDEAS TMG, NX SST, NX Advanced Flow, COMSOL and numerous R&D based CFD codes. Dr. Anderson has taught a graduate level CFD course several times at Cal Poly Pomona and his Ph.D. thesis involved direct numerical simulation of turbulent non-premixed flames. In addition, Dr. Anderson has published several refereed journal articles regarding CFD, Numerical Heat Transfer and testing of thermal hardware. Dr. Anderson can be contacted at kranderson1@csupomona.edu or Kevin.r.anderson@jpl.nasa.gov.

ACRONYMS

COTS = Commercial Off The Shelf

CFD = Computational Fluid Dynamics

OPALS = Optical Payload for Lasercomm Science

REFERENCES

1. "Heat Transfer" Incropera and Dewitt, 2nd. Ed., McGraw-Hill, 1990.
2. "Heat Transfer – A Practical Approach" by Y.A. Cengel, McGraw-Hill, 2003.
3. "Numerical Heat Transfer and Fluid Flow" by S. Patankar, Taylor & Francis, 1980.