



Computational Fluid Dynamics (CFD) Analysis of Optical Payload for Lasercomm Science (OPALS)

Sealed Enclosure Module

by

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

*Faculty Part-Time CFD Thermal/Fluids Analysis Engineer,
Thermal and Fluids Systems Engineering Group
NASA Jet Propulsion Laboratory, California Institute of Technology
and

Professor of Mechanical Engineering
California State Polytechnic University at Pomona

**OPALS Thermal Systems Engineer,
NASA Jet Propulsion Laboratory, California Institute of Technology

**OPALS Structural Cognizant Engineer,
NASA Jet Propulsion Laboratory, California Institute of Technology

Presented By

Dr. Kevin R. Anderson

Thermal & Fluids Analysis Workshop
TFAWS 2012

August 13-17, 2012

Jet Propulsion Laboratory,
California Institute of Technology
Pasadena, CA

© 2012 California Institute of Technology.
Government sponsorship acknowledged.





- Introduction
- Design Approach
 - Guidelines for Selecting Fans for Electronics Cooling
 - CFD Modeling Methodology
 - Design Cases Simulated
 - Worst case hot steady state
 - Transient cool-down
 - Thermal Engineering Unit Thermal Test Data Simulation / Model Correlation
 - On-station flow blockage / harness / strap simulation
 - Survival heater sizing design study
 - Thermal Desktop systems level SINDA model h-value correlation using CFD results
- Conclusions



Introduction



- This presentation documents the CFD analysis for a Commercial Off the Shelf (COTS) driven space mission which involved the use of forced convection cooling as the basis of its active thermal control 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
 - To this end, a close synergy between the CogE's and the CFD analyst is required in order to meet the project deadlines/requirements
- The results of CFD provide valuable flow-field visualization feedback on the validity of a proposed flow-tailoring geometry design



Introduction



- Per Incropera and Dewitt¹, handbook correlations for the convective film coefficient “h-value”, can differ from empirical test data as much as 25%
 - 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
- 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 thermal models of the overall hardware
 - This allows one to correlate the thermal model to actual on-station predictions with a higher degree of certainty
- CFD can be used to mitigate some of the risks/costs associated with determining the “h-values”



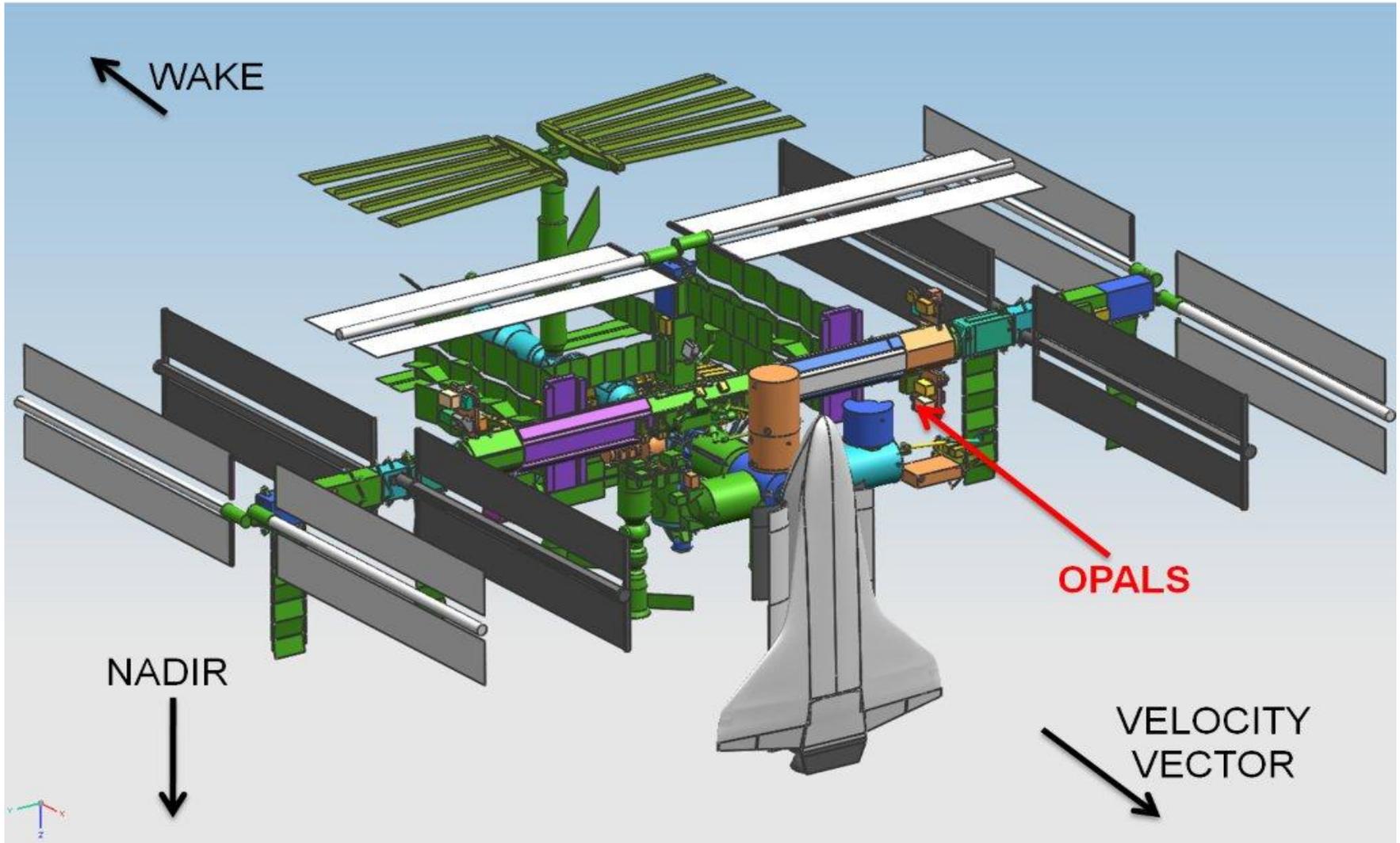
Design Approach



- The Optical Payload for Lasercomm Science (OPALS) mission for use on the International Space Station (ISS) was a JPL Early Career Hire (ECH) program
- Mentors in the form of senior engineers guided a team of Cognizant Engineers (CogE's) with 3 to 4 years experience typically
- Both a senior level CFD analyst and Stress Analyst were engaged to perform detailed modeling for the CogE's of the OPALS program
 - The analysts also served in the capacity of mentor, and had to form a close working relationship with the CogE's in order to meet the often times stringent demands of schedule and cost mandated by the project
- The scope and hardware of the OPALS mission is shown on the following charts



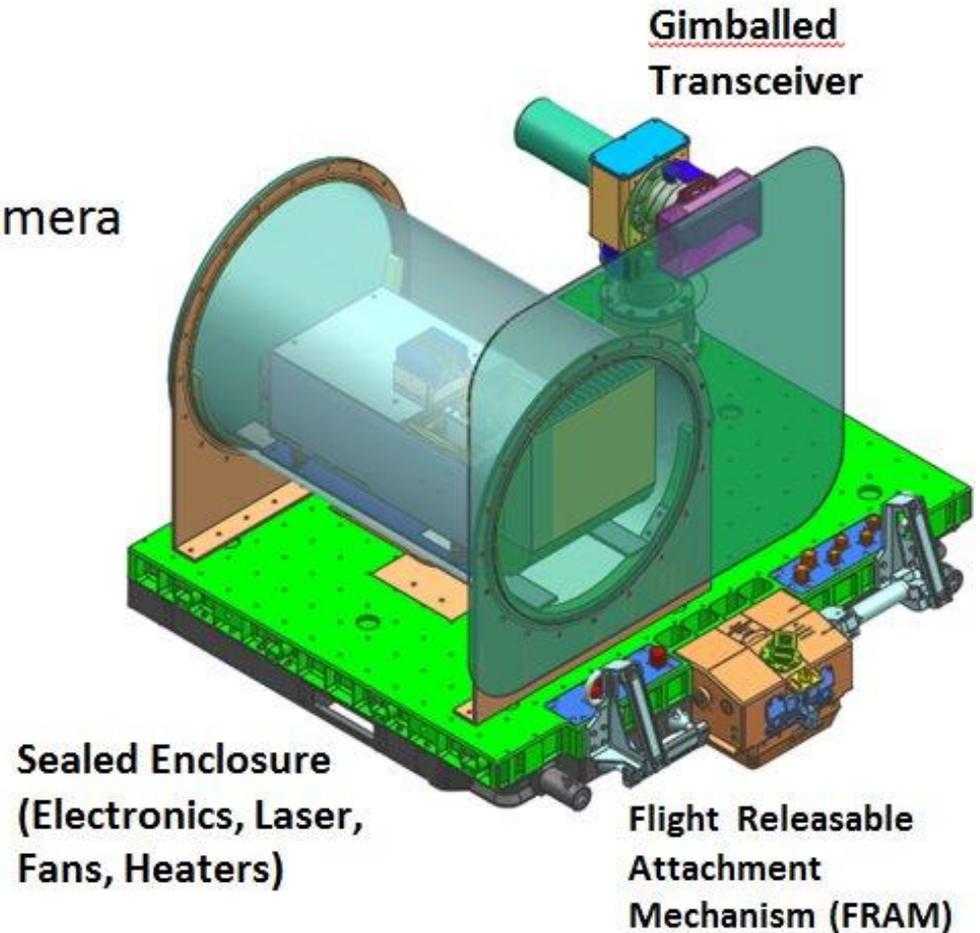
OPALS payload on the ISS





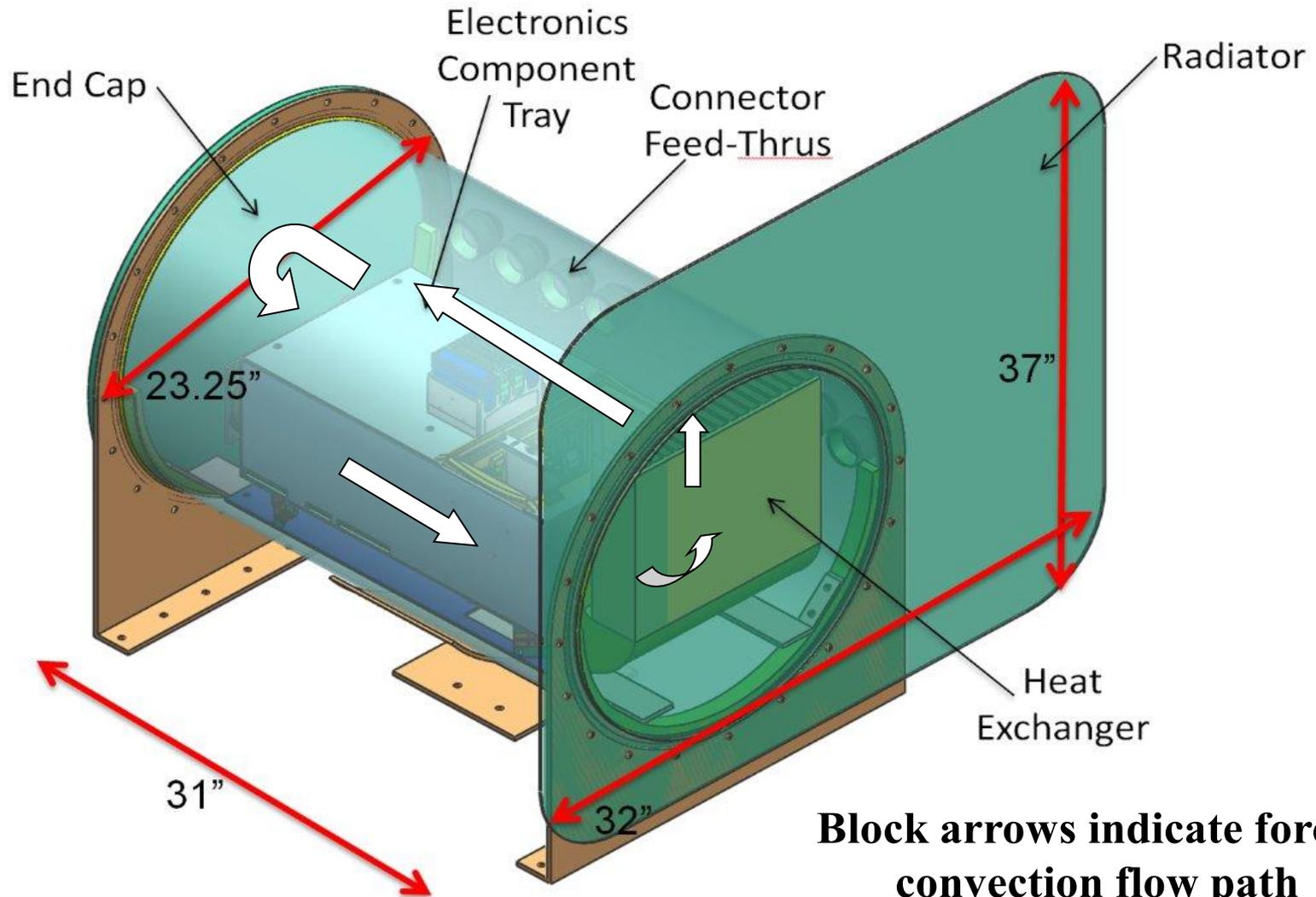
Flight System

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



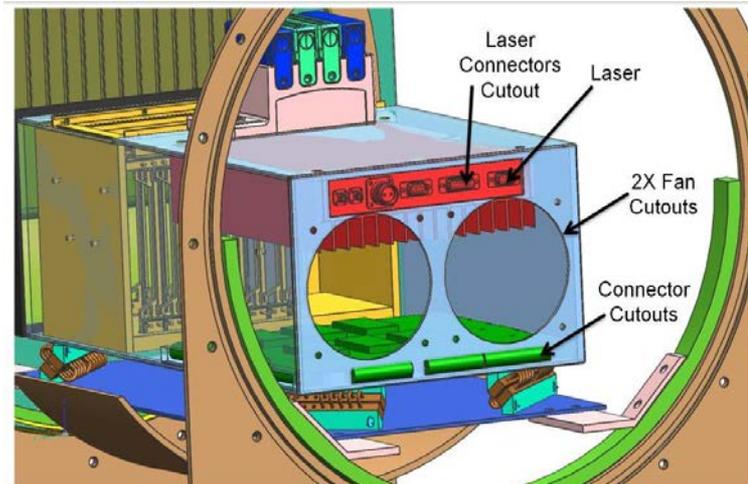
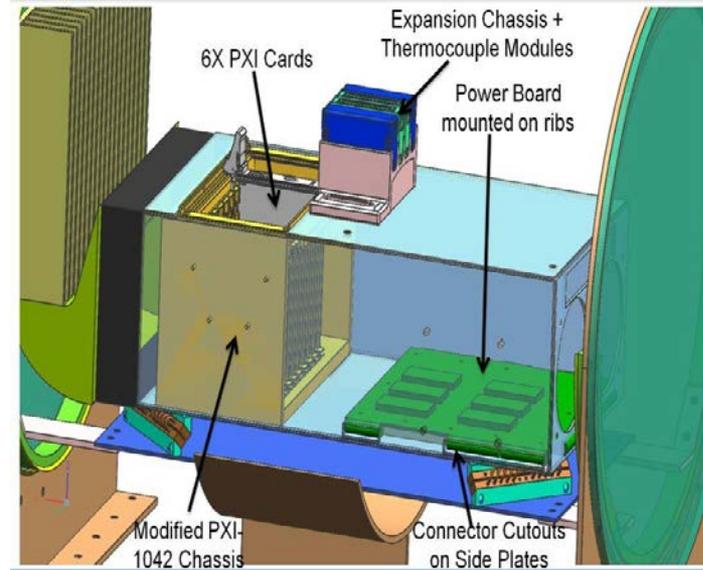
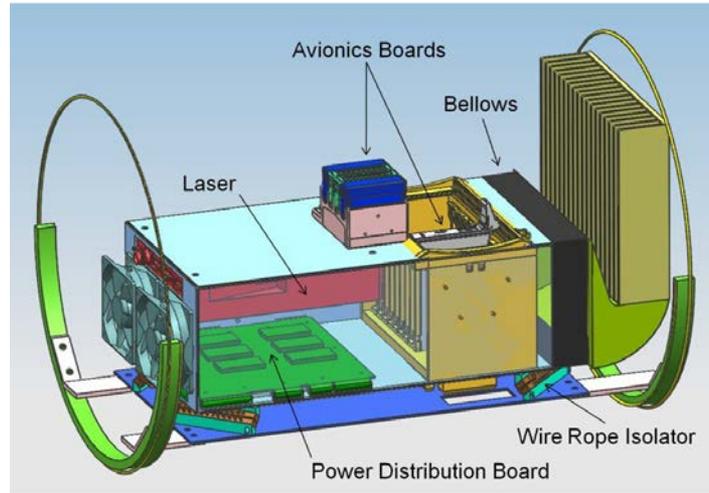


OPALS sealed enclosure sub-system





Sealed enclosure internal components





Design Approach



- The forced convection analysis focused on the sealed enclosure
- As seen by the hardware, the flow circuit involved a very complicated labyrinth of hardware
- The primary active thermal control system is comprised of the following components
 - heat exchanger
 - radiator
 - component tray
 - the component tray houses the various COTS avionics boards, as well as the power board electronics unit
 - duct/bellows
 - cooling fans
 - COTS cooling fans were used rated at 120 cfm and have 4 inch diameter blade



Design Approach



- The heat exchanger, radiators and duct are custom manufactured items
 - the heat exchanger (HX) 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
 - Intensive unit level CFD was performed on the HX at the early stages of this project
- 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
 - cool inlet air flows over the avionics and power board, picking up the dissipated heat via convection
 - this hot air is passed over the heat exchanger which is tied to the radiator
 - the heat liberated by the heat exchanger is then radiated via the radiator



Design Approach



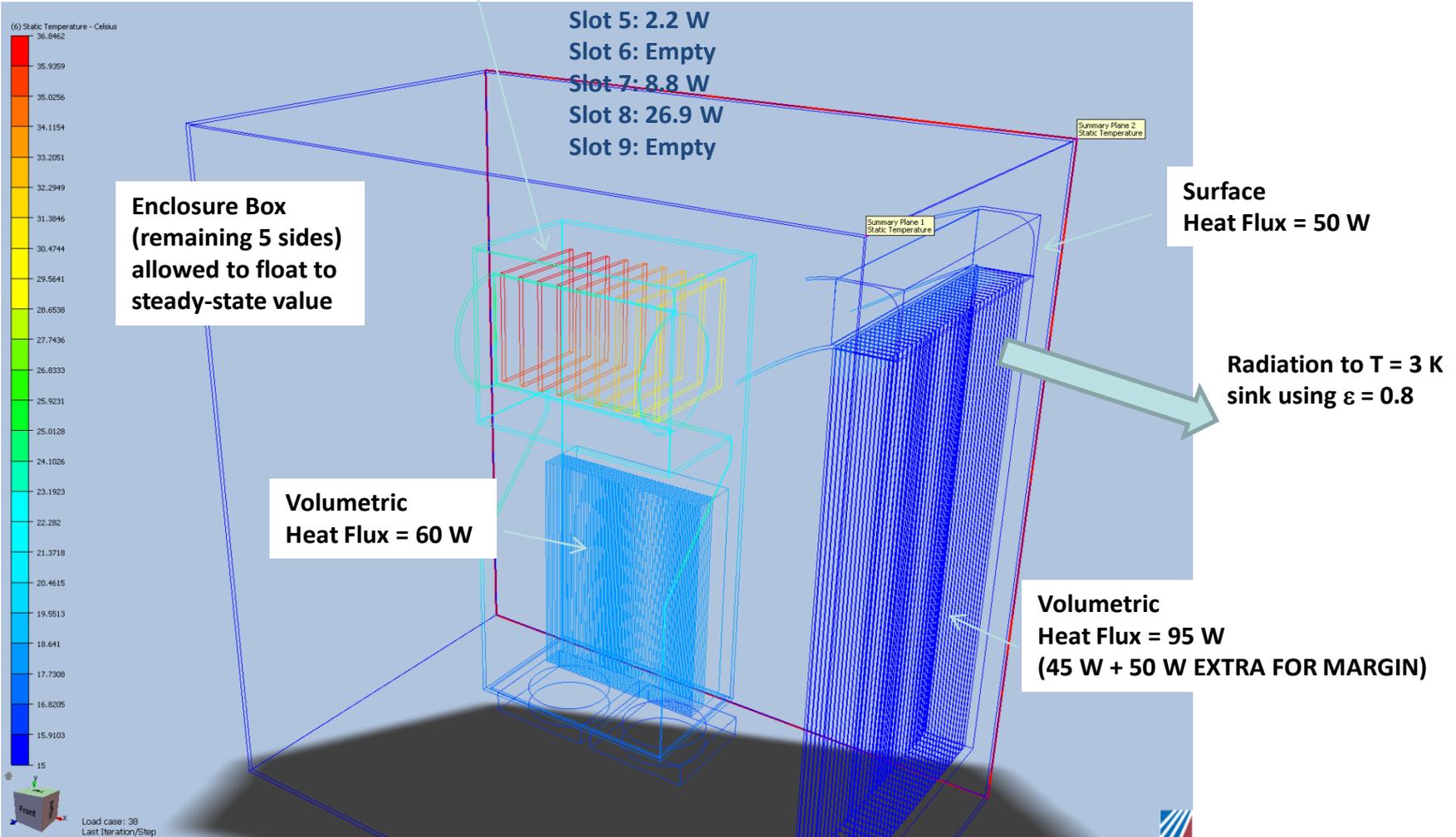
- For comparison purposes, the original concept for this design is shown on the next chart
 - the flow design matured greatly during the development of the final flight configuration hardware
- Along the way, several improvements were made in the flow path of the thermal control system
 - the final design virtually eliminates any drastic 90 degree turns in the flow field (with the exception of the interface between the duct and the heat exchanger)
 - 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 HX unit which could be housed within the cylindrical pressure vessel



Preliminary Design Concept

**Slot 1 -
Slot 9**

- Slot 1: 4.6 W
- Slot 2: Empty
- Slot 3: 1 W
- Slot 4: 12.9 W
- Slot 5: 2.2 W
- Slot 6: Empty
- Slot 7: 8.8 W
- Slot 8: 26.9 W
- Slot 9: Empty





Design Approach



- 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 and CPU intensive CFD runs
- Various thermal design guidelines per Cengel ² were adhered to when selecting the fans and sizing the flow path associated with electronics cooling
 - These guidelines are summarized on the following chart



Thermal design guidelines for using fans



Guideline Number	Guideline
1	CHECK IF NATRUAL CONVECTION IS SUFFICIENT
2	SELECT A FAN NEITHER TOO SMALL OR TOO LARGE UNDERSIZED = OVERHEATING OVERSIZED = MORE EXPENSIVE AND CONSUMES MORE POWER
3	MOUNT THE FAN AT THE INLET OF THE BOX AND FILTER THE AIR TO KEEP DIRT/DUST/PARTICULATES 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/AVIONICS SUCH THAT FLOW RESISTANCE IN THE BOX IS MINIMIZED



Thermal design guidelines for using fans



Guideline Number	Guideline
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/CARDS VERTICALLY AND BLOW THE AIR FROM BOTTOM TO TOP
9	AVOID FLOW SECTIONS WHICH INCREASE THE FLOW RESISTANCE I.E. AVOID UNNECESSARY CORNERS, SHARP TURNS, SUDDEN EXPANSIONS, AND/OR CONTRACTIONS, AND VERY HIGH VELOCITIES (> 7 m/s) SINCE THE FLOW RESISTANCE IS PROPORTIONAL TO THE FLOW RATE, AVOID VERY LOW VELOCITIES, SINCE THEY RESULT IN POOR HEAT TRANSFER AND ALLOW DUST/DIRT IN THE AIR TO SETTLE ON THE COMPONENTS



Thermal design guidelines for using fans



Guideline Number	Guideline
10	<p>FOR USE OF TWO OR MORE FANS, DECIDE WHETHER TO MOUNT THE FANS IN SERIES OR IN PARALLEL</p> <p>FANS IN SERIES WILL BOOST THE PRESSURE HEAD, THUS BEST FOR SYSTEMS WITH A HIGH FLOW RESISTANCE</p> <p>FANS IN PARALLEL WILL INCREASE THE FLOW RATE, THUS BEST FOR SYSTEMS WITH SMALL FLOW RESISTANCE</p>
11	<p>USE CFD TO EXERCISE 1. THROUGH 10. ABOVE (THIS GUIDELINE HAS BEEN ADDED BY THE PRIMARY AUTHOR OF THE PRESENT PRESENTATION)</p>
12	<p>PERFORM AN ENGINEERING UNIT LEVEL TEST TO CORRELATE THE CFD MODEL (THIS GUIDELINE HAS BEEN ADDED BY THE PRIMARY AUTHOR OF THE PRESENT PRESENTATION)</p>



- This CFD code used for this analysis was Blue Ridge Numeric's CFDDesign 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- ϵ turbulent closure model
 - The CFDDesign code uses the Galerkin FEM weighted residual method with pressure correction via the SIMPLE algorithm of Patankar³ to formulate and solve the equations of motion
 - CFDDesign is an affordable, turn-key, user friendly tool, and has a short learning curve, but as with all CFD tools it must be used with caution
 - All CFD tools have uncertainty, the job of the CFD analyst is to mitigate the level of this uncertainty by performing Verification and Validation (V&V) of the model being built and solved
 - Ultimately model correlation via test data is required to offer any real credibility or reliability to the CFD model



Modeling Workflow



- The modeling workflow steps are as follows:

Design Step	Description
1	Import CAD geometry as Parasolid
2	Create material data blocks CFDesign has a FAN object included in its material's database
3	Create B.C.'s / I.C.'s
4	Generate Mesh
5	Solve x, y, z momentum equations
6	Solve pressure correction equation
7	Correct velocities via pressure correction
8	Solve energy equation
9	Solve Turbulent Kinetic Energy equation
10	Solve Turbulent Kinetic Energy dissipation equation



Modeling Workflow



Design Step	Description
11	Check convergence (goto 5)
12	Perform output calculations
13	Write out data
14	Perform post-processing, i.e. plot h-values, etc.



Design Assumptions



- Various scenarios were simulated in this investigation, each orchestrated by the Thermal CogE and communicated to the CFD analyst
- 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 included
 - ISS-provided thermal environment definition
 - Stacked worst-case assumptions serve to bound environmental loads
 - With regards to the flow design the following methodology was adopted
 - Single flow path with major heat sources in series
 - Dry air (vs. He) selected on basis of cost and practicality



Design Assumptions



- Insulated interface between electronics and structure
- Single heat exchanger, strongly-coupled to radiator
- National Instruments hardware, designed for use in a room temperature environment with fan cooling
- 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*
- Several design cases were simulated with the CFD package in this study (see next chart)
 - 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

Design Case	Description
1	Worst case steady state hot
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



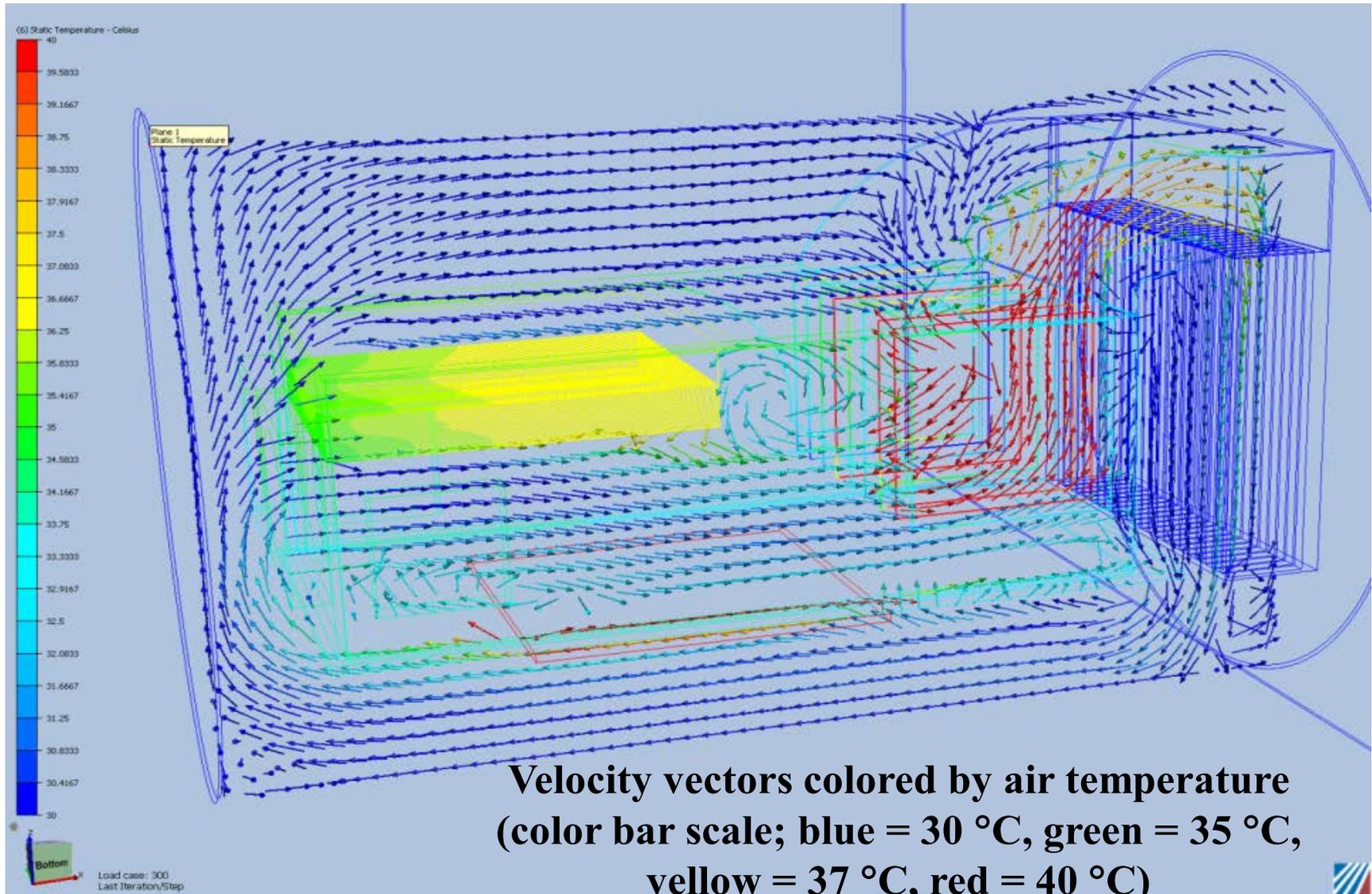
Worst case steady state hot



- This CFD simulation was performed to assess the on-station steady state performance of the sealed enclosure's thermal control system
- The following boundary conditions were used
 - radiation sink from radiator to 3 K using $\epsilon=0.8$
 - environmental flux of 239 W/m^2 applied to radiator
 - heat loads
 - cards = $\sim 60 \text{ W}$
 - laser = 60 W
 - power board = 44 W
 - Initial conditions of the air were $10 \text{ }^\circ\text{C}$, 14.7 psia .
 - The fan speed was 60 CFM per fan (other runs used 90 cfm)
 - 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 of $h = 30 \text{ W/m}^2\text{-K}$ on the heat exchanger

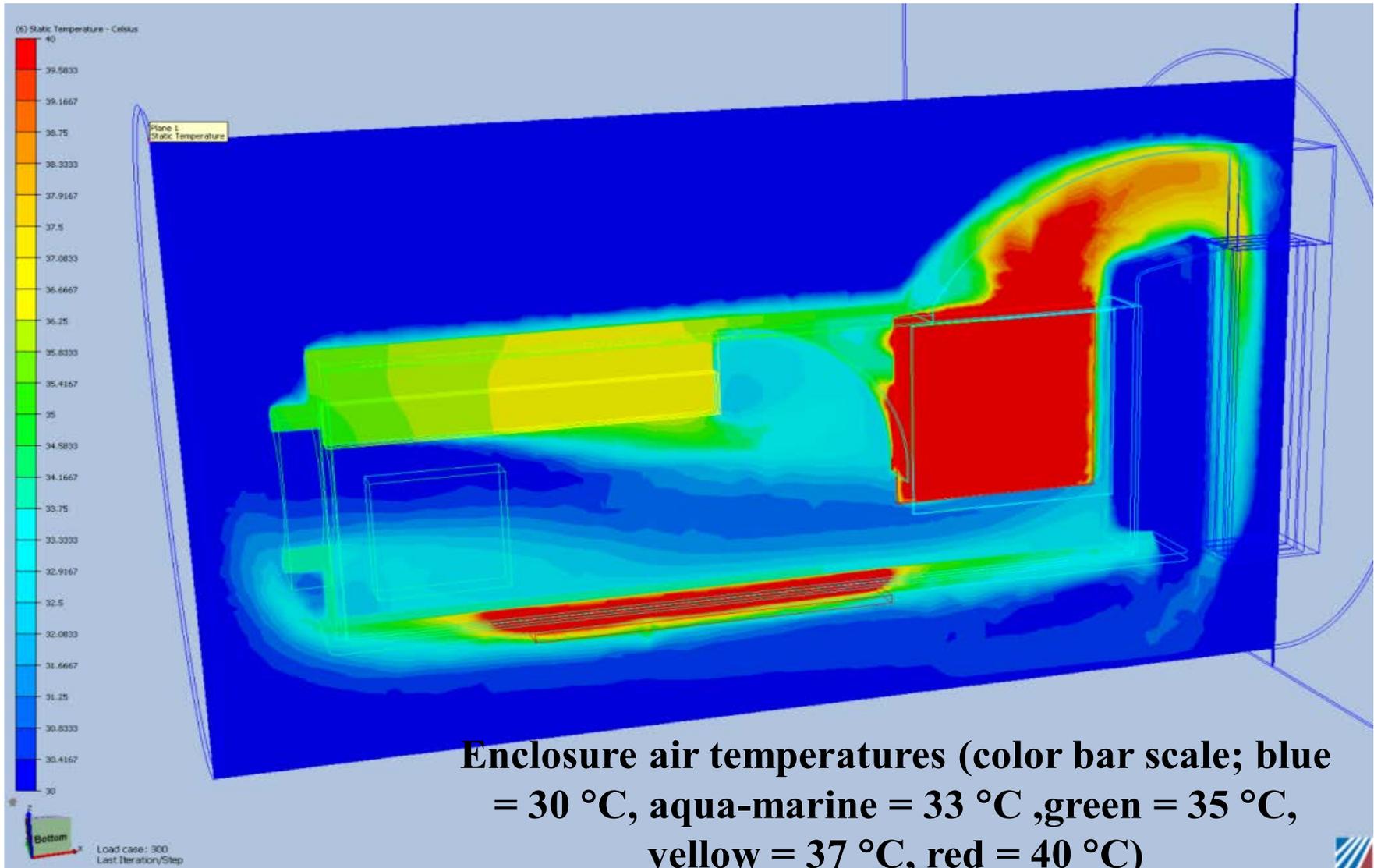


Worst case hot CFD simulation



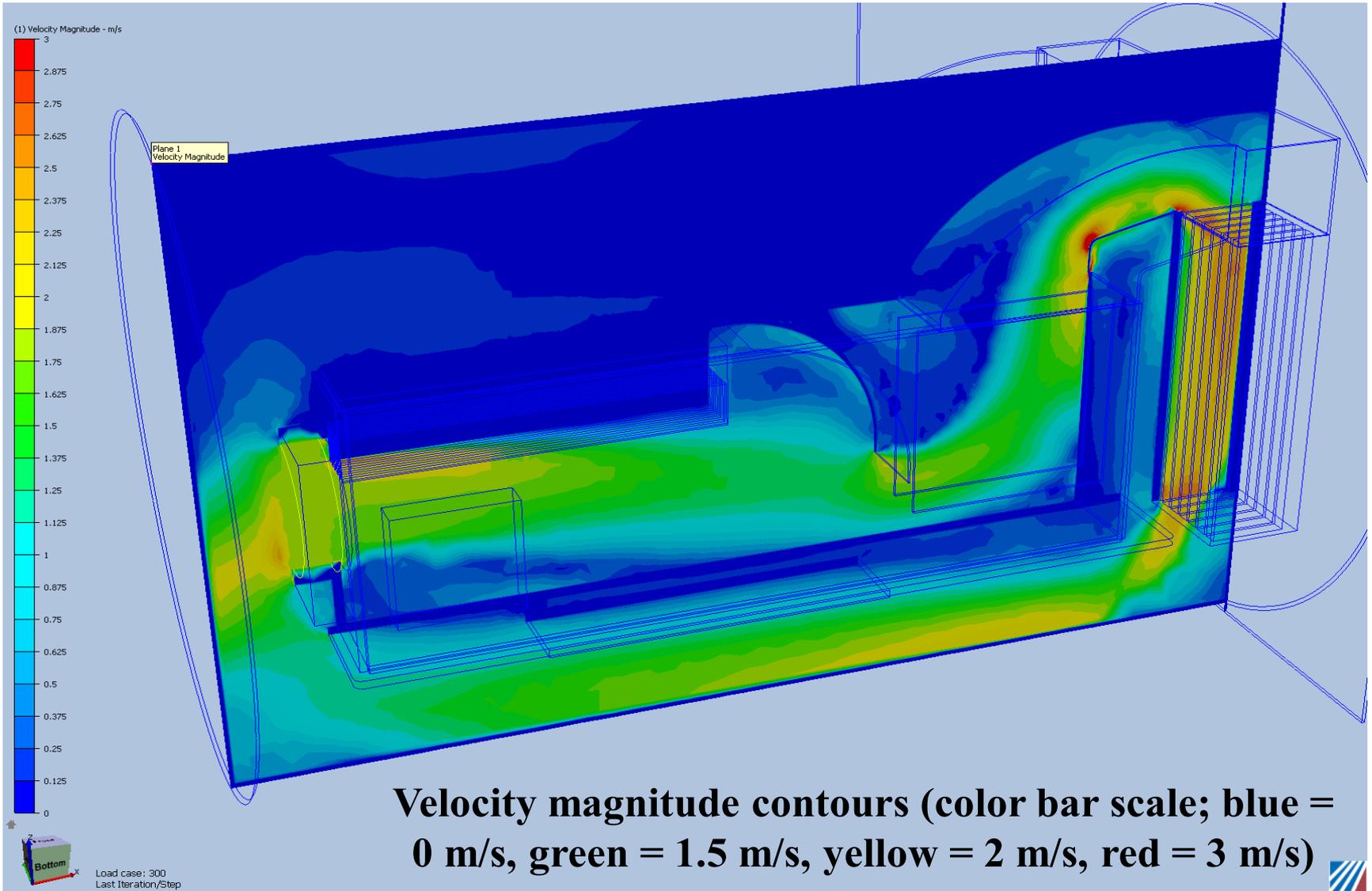


Worst case hot CFD simulation



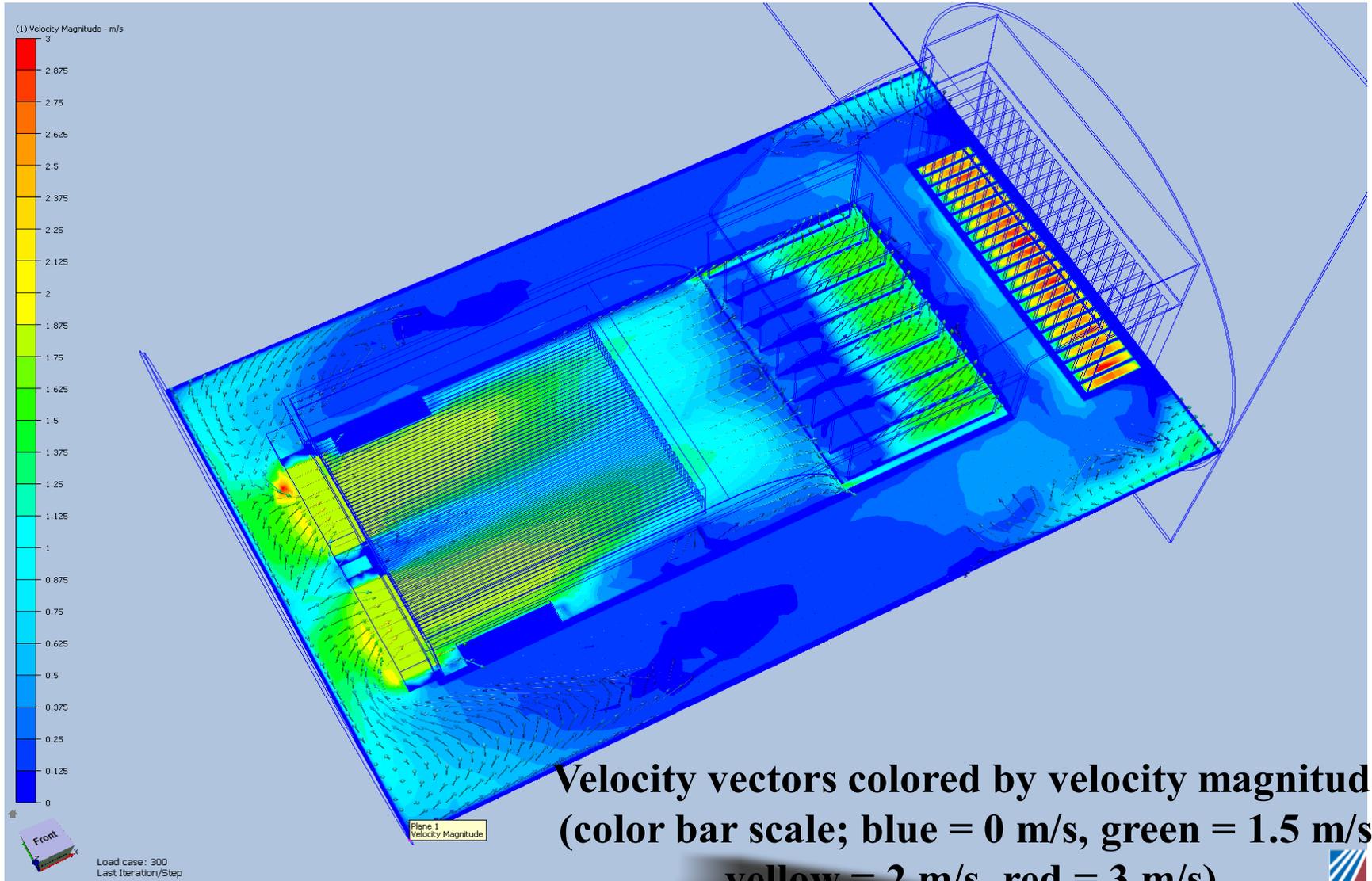


Worst case hot CFD simulation





Worst case hot CFD simulation





Worst case hot CFD simulation



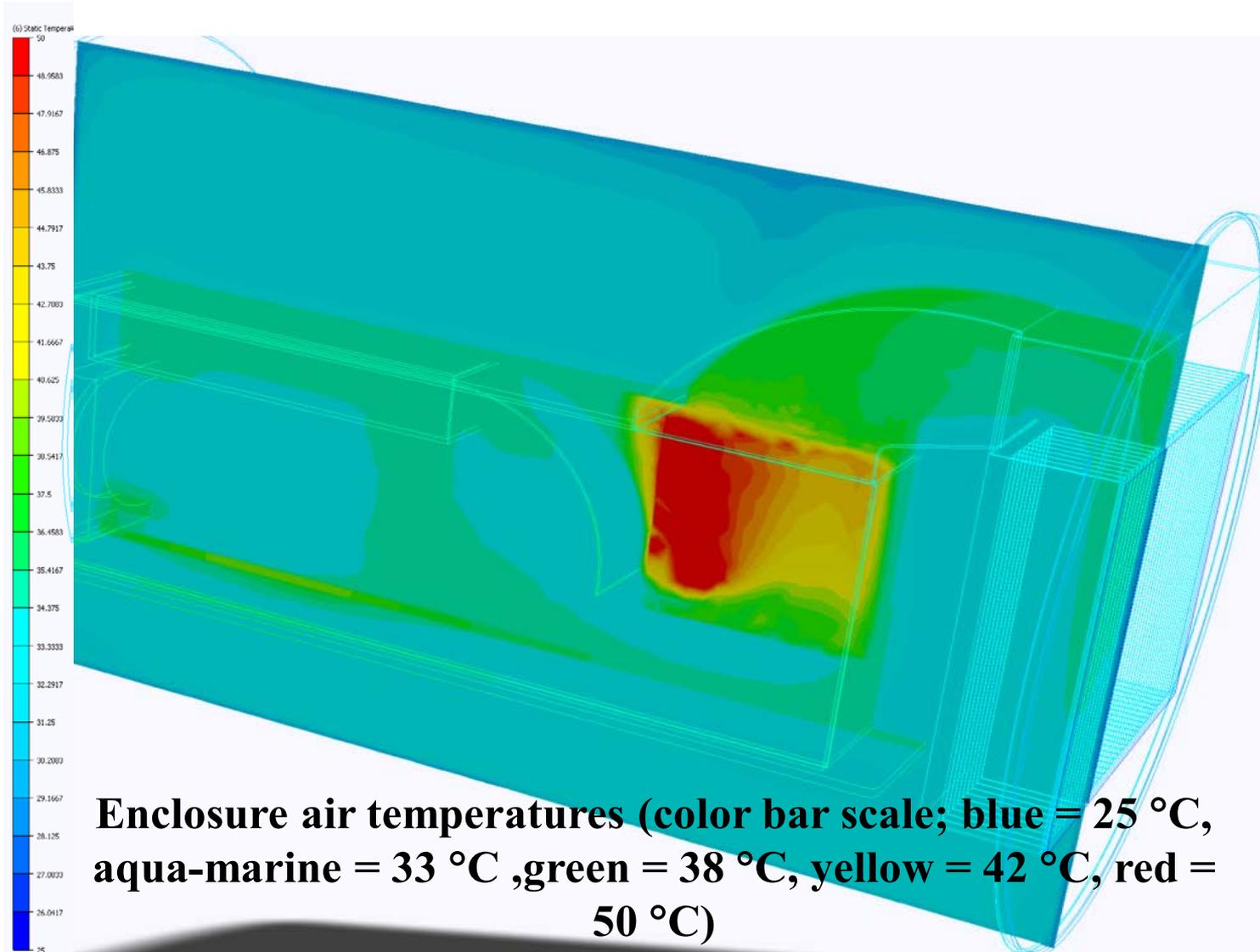
- The hottest air temperature is 40 °C near the card inlets
 - This satisfies the primary design requirement of having the air at the card inlet less than 50 °C
- Primary flow is confined to the “fan/duct/heat exchanger” network as engineered
- The velocity vectors indicate the draw-through which the fans entrain the surrounding fluid in the sealed enclosure
 - As this colder air is drawn into the duct the cooling system can perform the heat removal process again and repeat the desired cycle
- Pressure drop calculations showed that the dynamic pressure was on average one order of magnitude smaller than the static pressure
 - Hence friction factor and minor losses are matched with this size/type/gpm level of fan



- This CFD simulation was carried out in order to correlated the on-flight prediction CFD model against the Engineering Unit Test Mock-up
 - Typically, an Engineering unit test is mandatory when dealing with forced convection dominated flows
 - This is due to the uncertainty related to the “h-values”, which are non-deterministic by nature
 - Non-heritage and lack of internal databases also leads to large uncertainties when dealing with “h-values”
 - 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 test geometry was modeled in the CFD software
 - Results of the CFD Engineering Unit Test simulation were then compared with thermocouple test data
 - Minimal adjustments were needed to match the internal enclosure temperatures from the CFD model to that of the test data

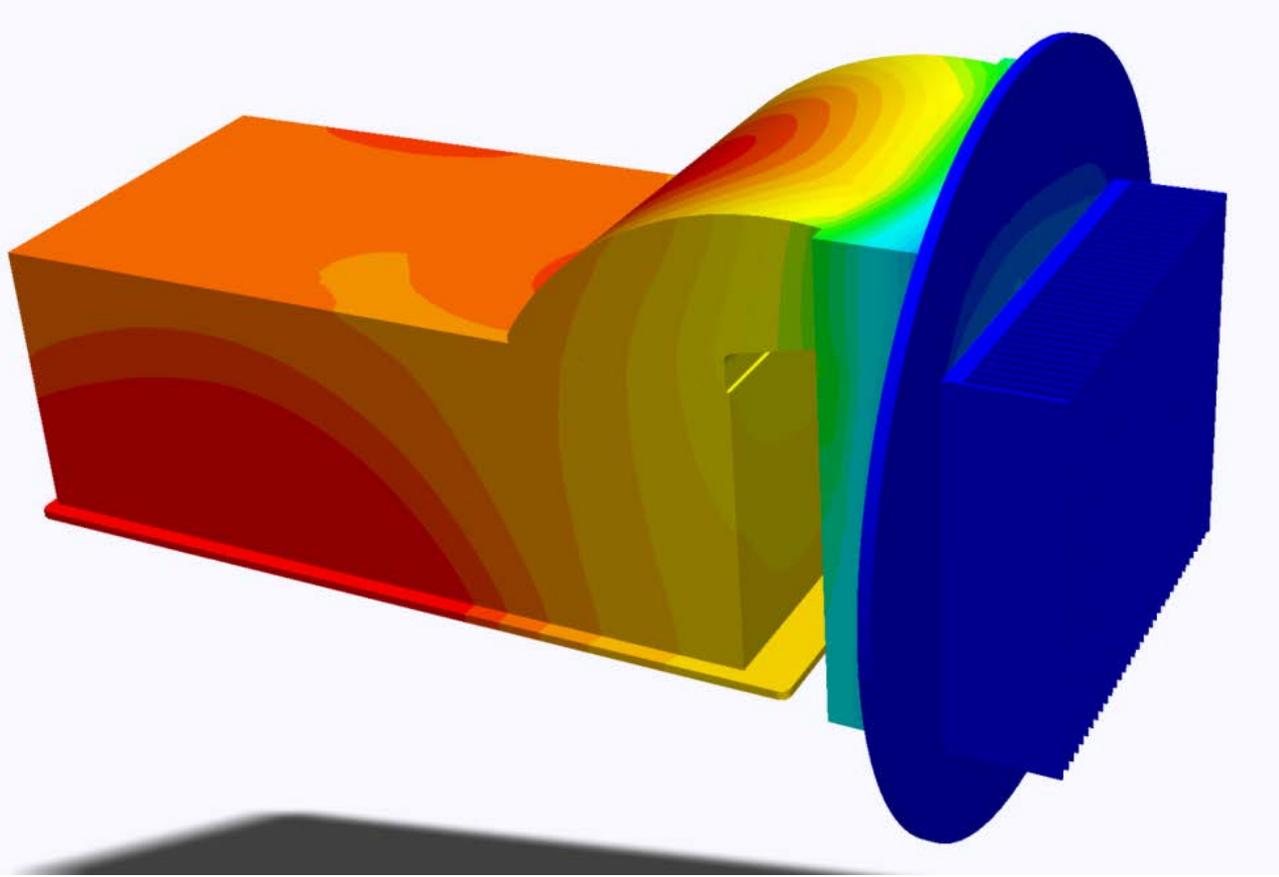


Engineering Unit Test Model Correlation





Engineering Unit Test Model Correlation



Component tray temperatures (color bar scale; blue = 32 °C, green = 33 °C, yellow = 34 °C, red = 35 °C)

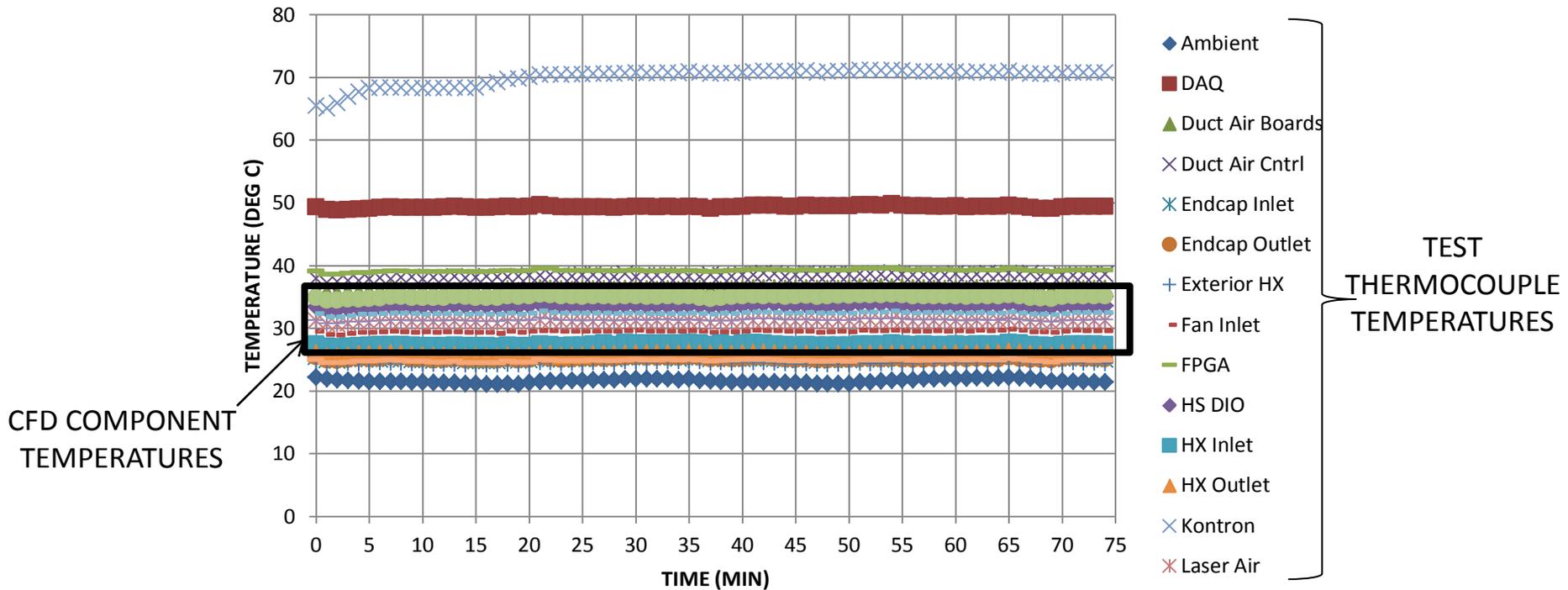


Engineering Unit Test Model Correlation



- CFD predicts 31 °C through 36 °C for components
- TC data reads 30°C through 35 °C for components
- Thus, the CFD model matches within 2 °C

TC DATA FOR SEALED ENCLOSURE TEST PEAK LOAD CASE





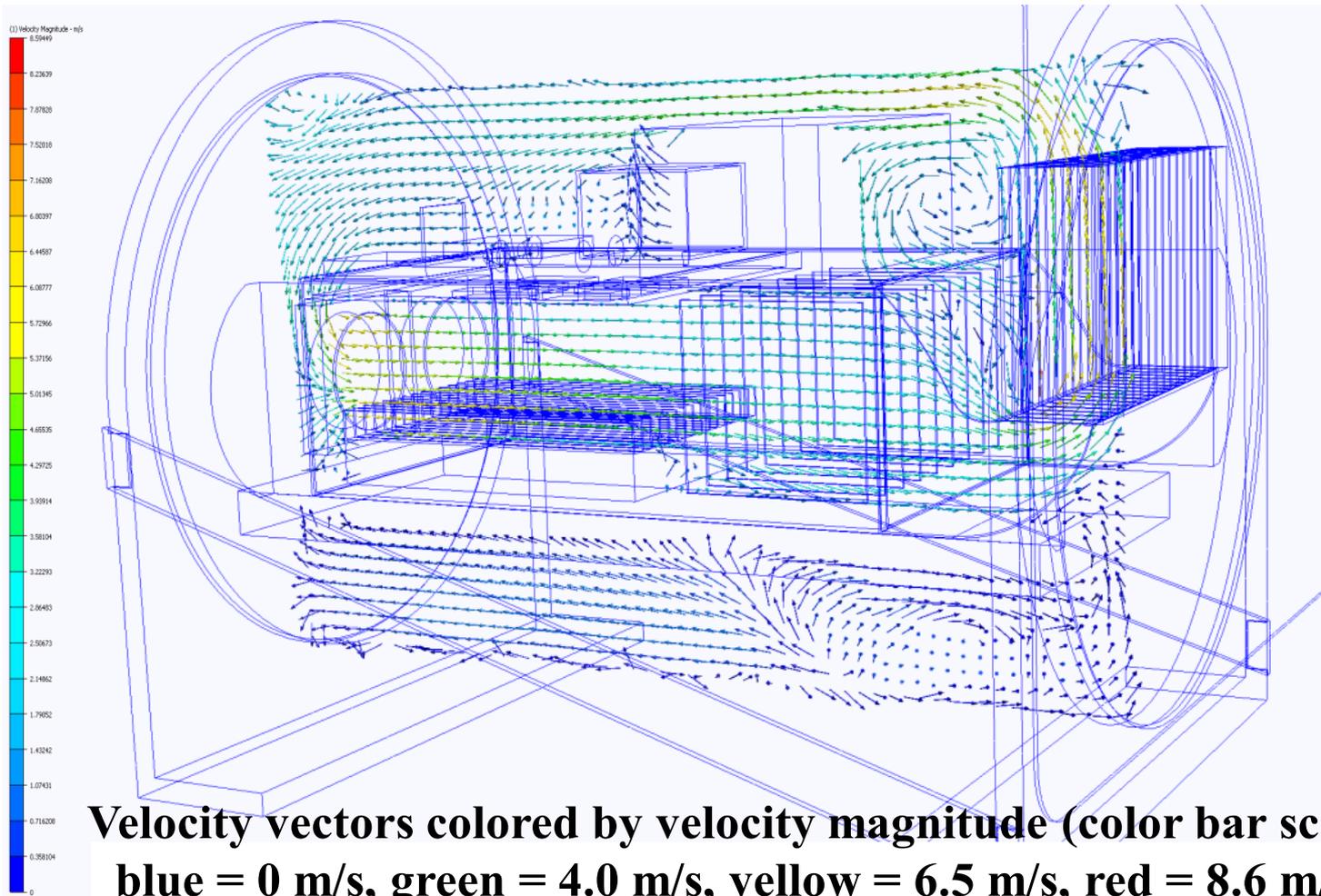
On-station flow blockage simulation



- Upon final assembly, the sealed enclosure incorporates several feed-throughs, and wire harnessing within the cylinder
 - This additional wiring and harnessing act as a flow impedance
- In order to mitigate risk, a 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 cylinder



On-station flow blockage simulation

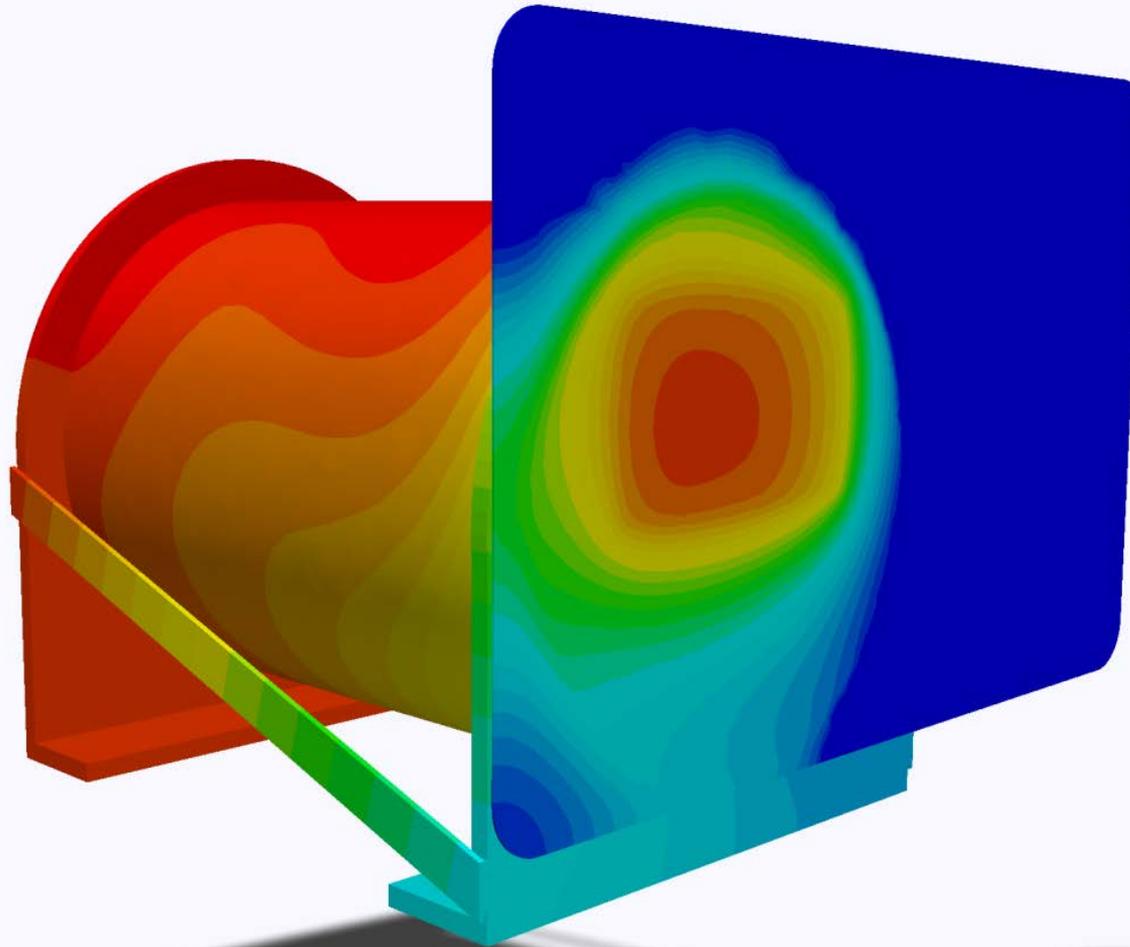




On-station flow blockage simulation



(6) Static Temperature - Celsius



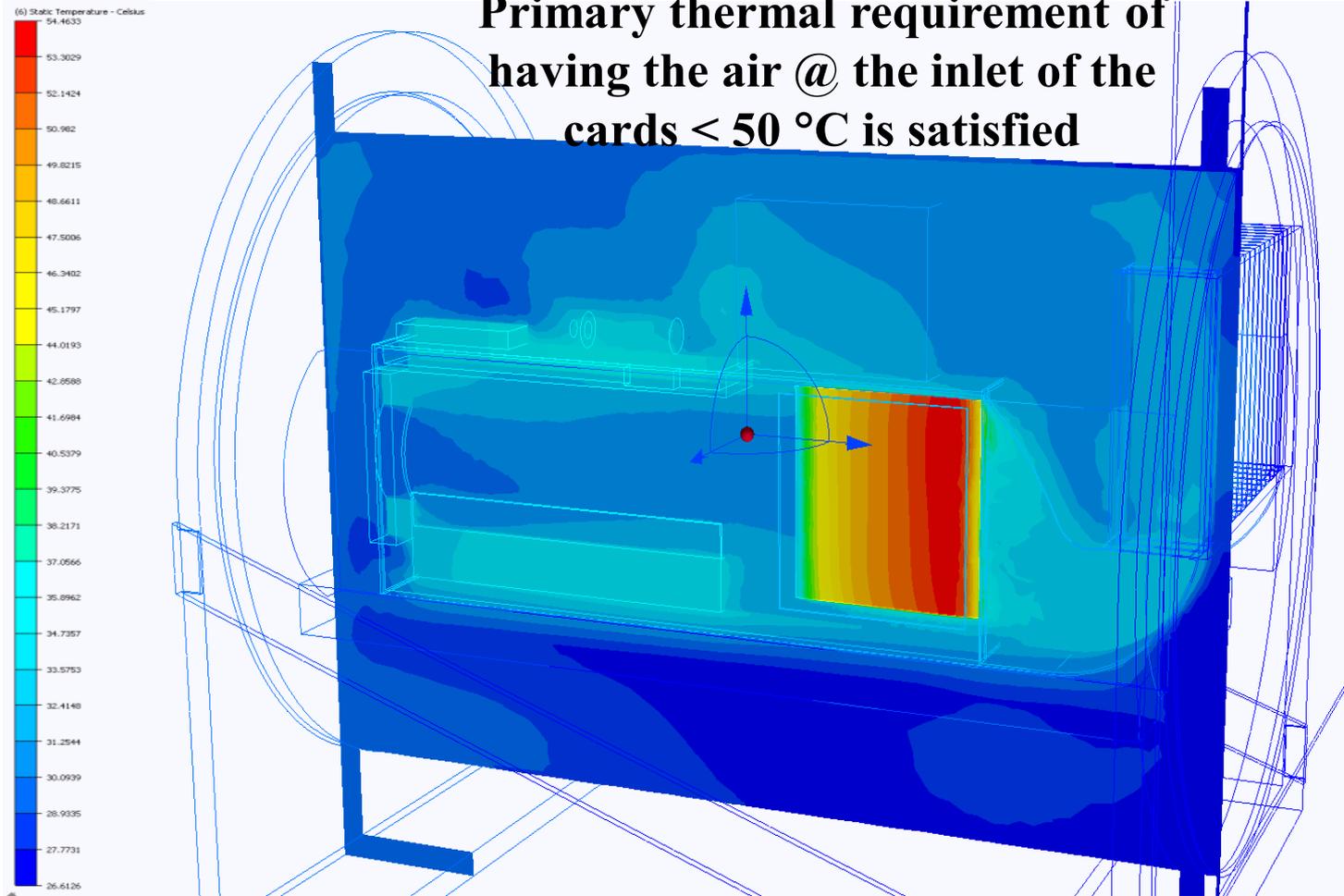
Cylinder, radiator and strap isotherms (color bar scale; blue = 18 °C, green = 24 °C, yellow = 27 °C, red = 30 °C)



On-station flow blockage simulation



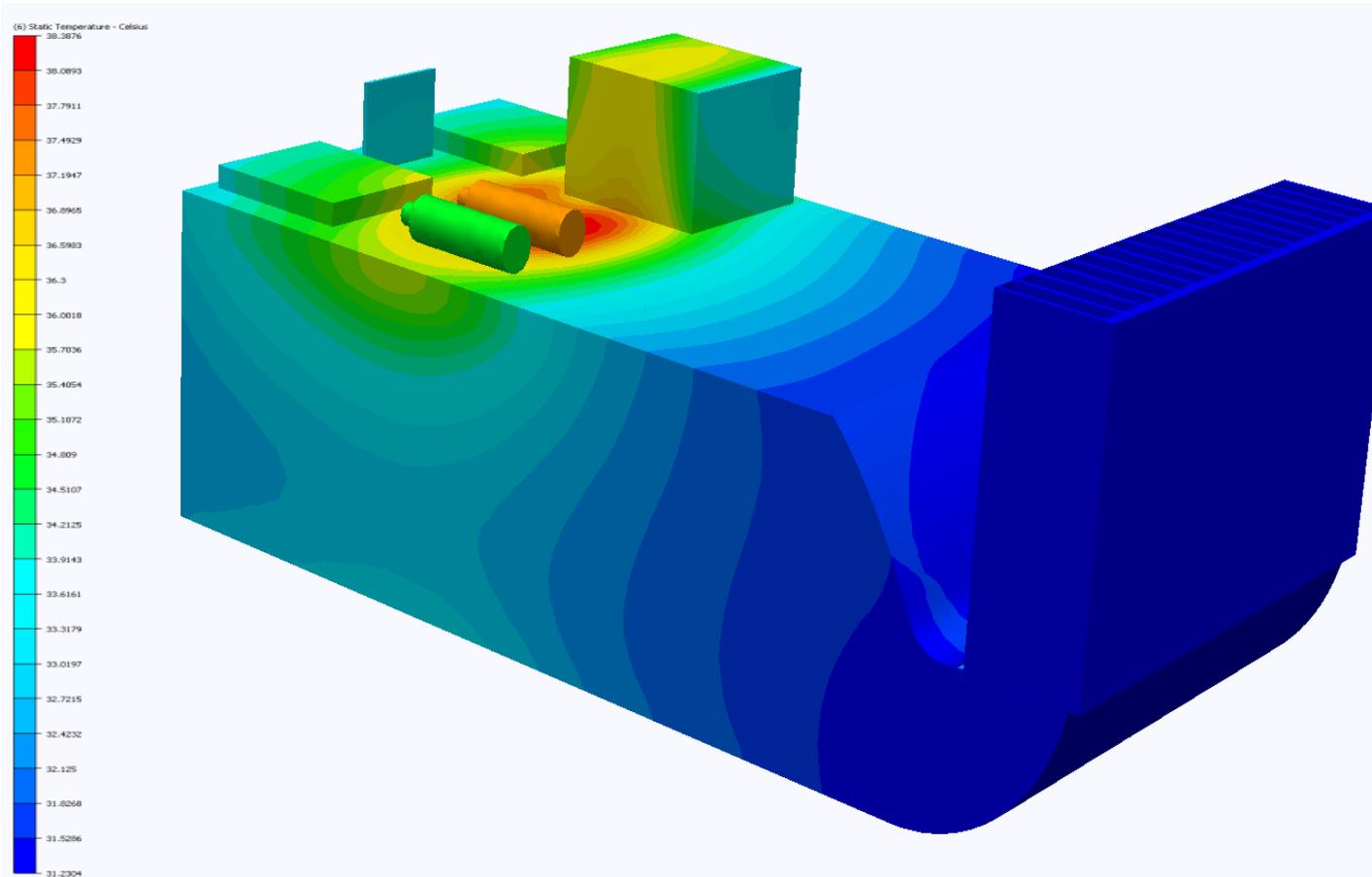
Primary thermal requirement of having the air @ the inlet of the cards $< 50\text{ }^{\circ}\text{C}$ is satisfied



Sealed enclosure air temperature in proximity of COTS NI cards (color bar scale; blue = $27\text{ }^{\circ}\text{C}$, green = $42\text{ }^{\circ}\text{C}$, yellow = $48\text{ }^{\circ}\text{C}$, red = $54\text{ }^{\circ}\text{C}$)



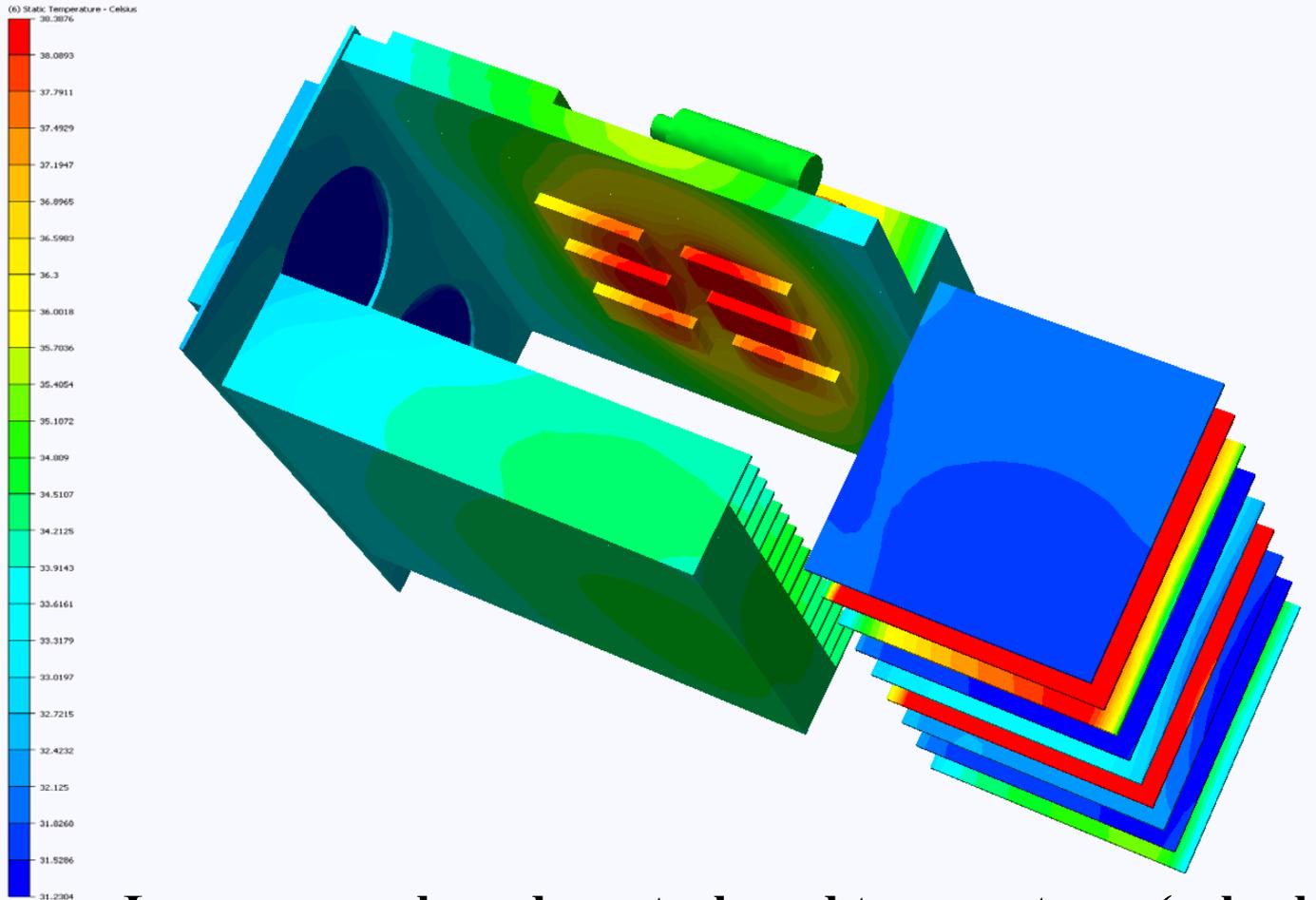
On-station flow blockage simulation



Component tray temperatures (color bar scale; blue = 31 °C, green = 35 °C, yellow = 37 °C, red = 38 °C)



On-station flow blockage simulation



Laser, power board, control card temperatures (color bar scale; blue = 31 °C, green = 35 °C, yellow = 36 °C, red = 38 °C).



On-station flow blockage simulation



- From the velocity vector fields, the inlet to the fans corresponds to average velocity ~ 5.2 m/s
 - For a 4 inch diameter fan leads to a average volumetric flow rate of 90 cfm (which matches the input)
- The thermal end-to-end gradient along the strap is found to be approximately 6 °C
- 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
 - For an interfacial contact conductance between the radiator and the heat exchanger of $h = 1500$ W/m²-K a drop of ~ 5 °C between the center of the heat exchanger and the radiator occurs, which is typical for a such gasketed/bolted thermal interface



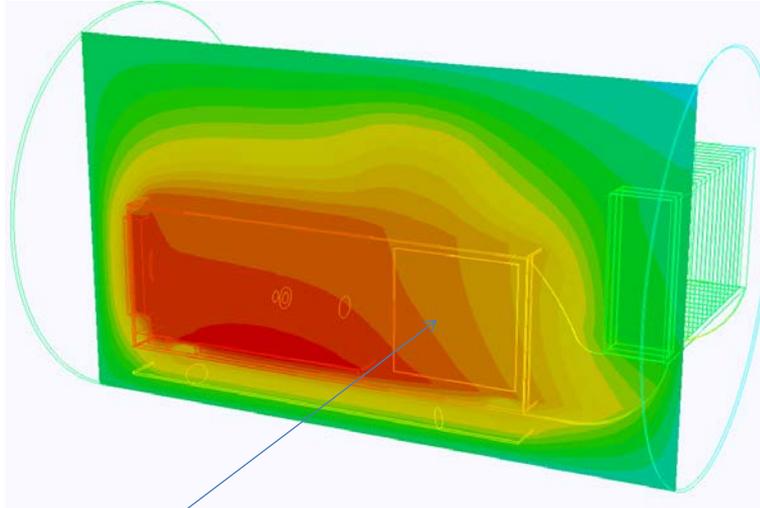
Survival heater sizing design study



- The CFD simulation was performed in order to assess the correct heater power rating for the survival heaters
- The results of this trade study guided the proper selection of the survival heaters, as well as the design of the thermostat algorithm for control of these heaters
- The following were the key simulation parameters
 - All electronics were turned off, including the fans
 - The air medium was modeled in CFD 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
 - Typical results of this trade are shown on the next chart



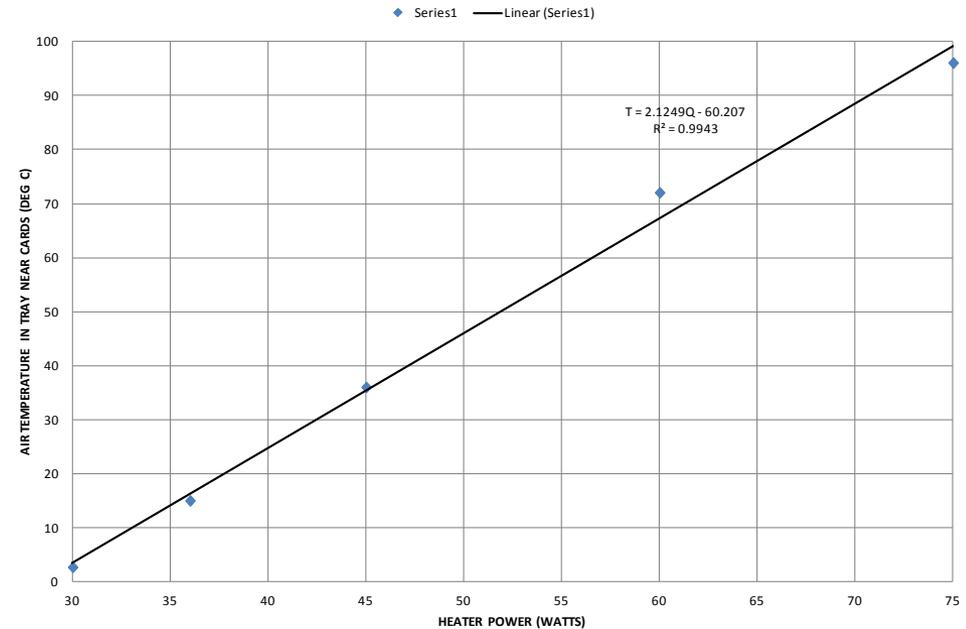
Survival heater sizing design study



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

**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)**

SURVIVAL HEATER SIZING TRADE STUDY FOR OPALS SEALED ENCLOSURE





SINDA h-value correlation using CFD



- 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
 - This SINDA model is used in the SPACE-X dragon launch vehicle Thermal Desktop model and also in the Thermal Desktop model of the ISS itself
- The internal “air nodes” of the SINDA model are connected by $G=h*A$ “convection conductors”, the values for h were taken from the CFD simulation and used in the SINDA network



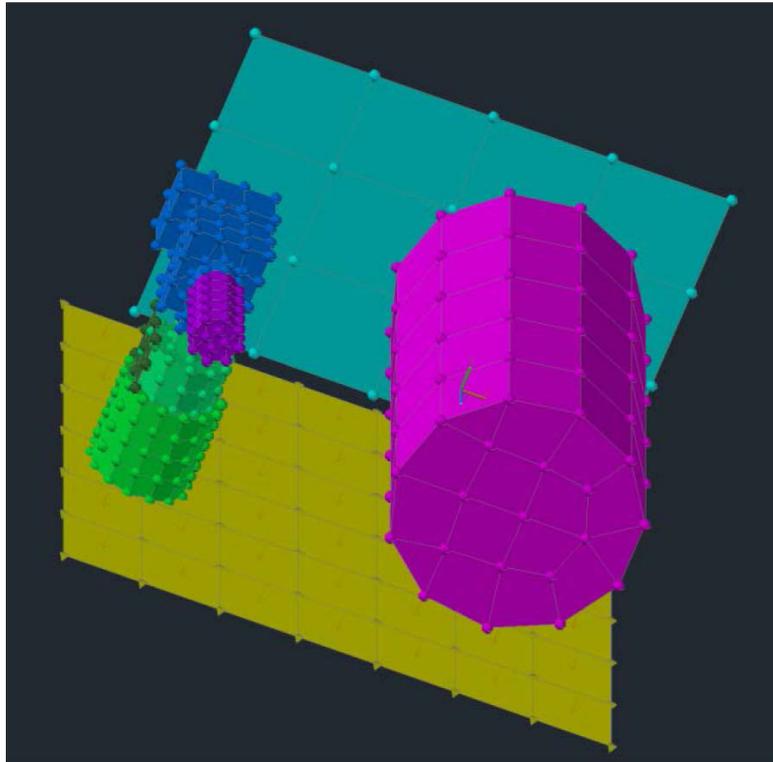
SINDA h-value correlation using CFD



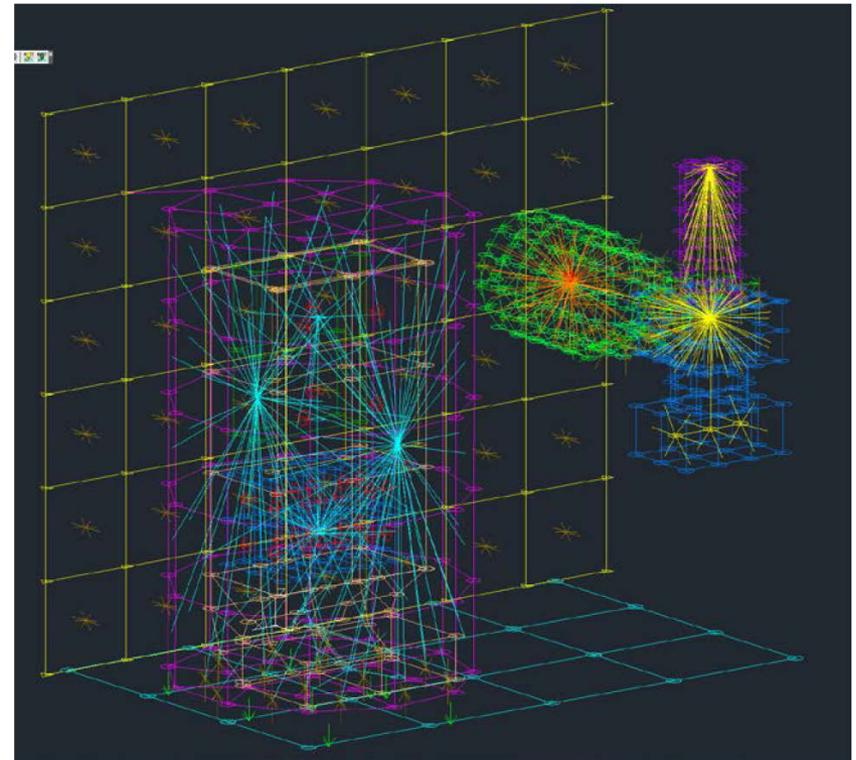
- The $G = h \cdot A$ conductors of the SINDA model tie the wall of the cylindrical sealed enclosure to the component tray, end caps, heat exchanger and duct, etc.
- Upon correlation of the SINDA model, typical values of h within the sealed enclosure ranged from a minimum of $7.5 \text{ W/m}^2\text{-K}$ (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 $h = 5 \text{ W/m}^2\text{-K}$ per Incropera & Dewitt¹
 - The layout of the Thermal Desktop model is shown on the next chart
 - The $G = h \cdot A$ conductors are shown within the sealed enclosure



SINDA h-value correlation using CFD



**Thermal Desktop
Systems Level
OPALS
Thermal Model**



**Internal $G = hA$ “air conductors”
shown in cyan**



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 short turn-around times
 - Several simulations witnessed during the evolution and CFD analysis of a forced “convective driven” active thermal control system design have been presented herein
- The results of this presentation have demonstrated the wide range of thermal/fluids sub-system component modeling scenarios to which CFD can be applied



Conclusions



- Obviously, turn-key CFD should be thought of as a design roadmap, and not a final solution
 - Due to their non-deterministic nature, theoretically derived forced convective turbulent flow heat transfer coefficients can have uncertainties from 25% to 50%.
 - Testing should ultimately be carried out whenever one encounters forced convection comprising the core of the thermal control system
 - The use of CFD can mitigate the uncertainties involved in predicting/modeling convective flows
 - The complexity and fidelity of desktop based, fast turn-around type of CFD modeling presented herein is a necessary avenue to clearly understand any design hurdles one may encounter along the path to development of a project of this caliber
 - This process requires close synergy between the CogE's and the CFD/Thermal analyst since it inevitably requires many time-consuming iterations which will strain time/budget driven projects



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



Biography



- Dr. Kevin R. Anderson, P.E. is a Faculty Part Time Sr. Thermal Engineer/CFD Analyst for NASA JPL's Thermal & 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 has acquired over 15 years of practical experience working in the aerospace engineering thermal sector
 - Dr. Anderson is conversant in several CFD packages including ANSYS-FLUENT, STAR-CCM, CFD2000, CFDDesign 2012, IDEAS TMG, NX SST, NX Advanced Flow, COMSOL and numerous R&D (FORTRAN or C based) turbulent combustion oriented CFD codes
 - Dr. Anderson regularly teaches a graduate level CFD course at Cal Poly Pomona and has published several refereed journal articles related to CFD & Numerical Heat Transfer
 - Dr. Anderson can be contacted at Kevin.r.anderson@jpl.nasa.gov



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