RECEIVED
NOV 02 1998
OSTI

*Thermal Issues Associated with the
Lighting Systems, Electronics Racks, and
Pre-Amplifier Modules in the
National Ignition Facility*

Los Alamos
NATIONAL LABORATORY

*Los Alamos National Laboratory is operated by the University of California
for the United States Department of Energy under contract W-7405-ENG-36.*

An Affirmative Action/Equal Opportunity Employer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither The Regents of the University of California, the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by The Regents of the University of California, the United States Government, or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of The Regents of the University of California, the United States Government, or any agency thereof. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.

DISCLAIMER

**Portions of this document may be illegible
electronic image products. Images are
produced from the best available original
document.**

*Thermal Issues Associated with the
Lighting Systems, Electronics Racks, and
Pre-Amplifier Modules in the
National Ignition Facility*

J. D. Bernardin

A. C. Owen

K. L. Lam

TABLE OF CONTENTS

Abstract	1
Nomenclature	2
1. Introduction	3
2. Light Fixture Evaluation	7
2.1 General Modeling Approach of a Fluorescent Light Fixture	7
2.1.1 Heat Dissipation Characteristics of a Fluorescent Light Fixture	7
2.1.2 CFD Code	9
2.2 Light Fixture Suspended from a Beam	9
2.2.1 Model Geometry, Boundary Conditions, and Grid Discretization	10
2.2.2 Convergence Criteria	12
2.2.3 Results/Design Recommendations	13
2.3 Light Fixture Mounted to a Wall, 2-D Model	20
2.3.1 Model Geometry, Boundary Conditions, and Grid Discretization	20
2.3.2 Results/Design Recommendations	23
2.4 Light Fixtures Mounted to a Wall, 3-D Model	29
2.4.1 Model Geometry, Boundary Conditions, and Grid Discretization	29
2.4.2 Results/Design Recommendations	33
2.5 Summary of Light Fixture Design Recommendations	34
3. Electronics Rack Evaluation	38
3.1 Heat Dissipation Characteristics of an Electronics Rack	38
3.2 Model Geometry, Boundary Conditions, and Grid Discretization	40
3.3 Convergence and Analytical Verification	46
3.4 Results	50
3.4.1 Linear Temperature Profile Boundary with No Downward Airflow	51
3.4.2 Heat Flux Boundary with No Forced Airflow	55
3.4.3 Heat Flux Boundary with Forced Airflow	61
3.5 Summary of Electronics Rack Design Recommendations	65
4. Pre-Amplifier Module (PAM) Evaluation	69
4.1 Heat Dissipation Characteristics of a PAM	69
4.2 Model Geometry, Boundary Conditions, and Grid Discretization	73

4.3 Results -----	87
4.3.1 Model A Results -----	88
4.3.2 Model B Results-----	97
4.4 Lumped-Parameter Analysis -----	103
4.5 Summary of PAM Design Recommendations-----	107
References -----	108
Appendix-----	109

**Thermal Issues Associated with the Lighting Systems, Electronics Racks,
and Pre-Amplifier Modules in the National Ignition Facility**

by

J. D. Bernardin, A. C. Owen, and K. L. Lam

Abstract

This report summarizes an investigation of the thermal issues related to the National Ignition Facility. The influence of heat sources such as lighting fixtures, electronics racks, and pre-amplifier modules (PAMs) on the operational performance of the laser guide beam tubes and optical alignment hardware in the NIF laser bays were investigated with experiments and numerical models. In particular, empirical heat transfer data was used to establish representative and meaningful boundary conditions and also serve as bench marks for computational fluid dynamics (CFD) models. Numerical models, constructed with a commercial CFD code, were developed to investigate the extent of thermal plumes and radiation heat transfer from the heat sources. From these studies, several design modifications were recommended, including reducing the size of all fluorescent lights in the NIF laser bays to single 32 W bulb fixtures, maintaining minimum separation distances between light fixtures/electronics racks and beam transport hardware, adding motion sensors in areas of the laser bay to control light fixture operation during maintenance procedures, properly cooling all electronics racks with air/water heat exchangers with heat losses greater than 25 W/rack to the NIF laser bay, ensuring that the electronics racks are not overcooled and thus maintain their surface temperatures to within a few degrees centigrade of the mean air temperature, and insulating the electronic bays and optical support structures on the PAMs.

NOMENCLATURE

Symbol	Description
A	Area
c_p	Specific heat
Gr	Grashoff number
g	Gravitational constant (9.18 m/s^2)
h	Heat transfer coefficient
k	Thermal conductivity
L	Characteristic length
m	mass flow rate
Nu	Nusselt number ($h L/k$)
Pr	Prandtl number ($c_p \mu/k$)
q''	Heat transfer rate per unit surface area
Ra	Rayleigh number
Re	Reynolds number ($\rho u l/\mu$)
T	Temperature
V	Velocity
x	Cartesian coordinate direction

Greek Symbol	Description
α	Thermal diffusivity
β	Linear coefficient of thermal expansion for a gas (see Equation (4))
ΔT	Temperature difference
μ	Dynamic viscosity
ρ	Density
ν	Kinematic viscosity

Subscript	Description
L	Parameter corresponding to characteristic length L
S	Surface
$Surr$	Surroundings
w	Wall
∞	Corresponding to environment

1. Introduction

This report summarizes an investigation of thermal issues related to the operation of the National Ignition Facility (NIF). In particular, the potential interference of waste heat from the lighting fixtures, electronics racks, and pre-amplifier modules (PAMs) is reviewed and discussed, as well as its impact on the operational performance of the laser beam transport tubes and optical alignment components. It has been recognized by NIF design engineers that spatial and temporal excursions in the NIF environmental air temperature by undesired heat loads may significantly degrade the operating performance and net yield of the NIF. A previous study by Bernardin et al. [1] revealed that temporal oscillations in the temperature of the air delivered from the HVAC system and spatial temperature variations caused by waste heat from light fixtures had a negative impact on the NIF beam transport system. Based upon that study, design recommendations were made to minimize the negative effects of the nonuniform temperature fields.

The current NIF design calls for a desired temperature field of $20.00 \pm 0.28^{\circ}\text{C}$ throughout the facility. Obviously, heat sources within the facility will generate temperatures outside of this range. The purpose of this report is to assess the extent of spatial temperature excursions created by heat sources in the NIF laser bays and offer design recommendations to minimize their impact on operating performance. Figure 1 conveys the approach used in this study to determine the region of detrimental influence of a heat source. First, experiments were performed to determine temperature and/or heat flux surface conditions on a particular heat source. Next, a computational fluid dynamics (CFD) model of the heat source and surrounding environment was generated. Using the experimentally determined temperature or heat flux boundary condition, the CFD model

was solved for the magnitude and extent of the thermal plume emanating from the heat source. By incorporating the results of the CFD modeling with facility layout drawings, it was determined which, if any, critical laser transport or alignment components were in close enough proximity of the heat source to be negatively affected by the thermal plume. If a potential interference was present, design modifications were recommended to alleviate or minimize the problem.

Three different thermal studies are discussed in this report. First, experimental temperature measurements and a discussion of the heat transfer characteristics of fluorescent light fixtures are presented. In addition, a detailed numerical study of the thermal plume characteristics from various light fixtures was performed using the commercial CFD software package CFX [2]. The extent of thermal plumes under various operating conditions, combined with an analytical radiation analysis [1], was used to determine minimum allowable separation distances between light fixtures and optical components and/or maximum power ratings for various light fixtures. The second study is associated with waste heat release from electronics racks. As with the light fixtures, experimental temperature measurements from an electronics rack are discussed to better understand its heat dissipation characteristics. Next, a two-dimensional CFX model of the electronics rack was used to generate temperature and velocity fields to study the impact of the heat loss on the surrounding environment over a wide range of operating conditions. The third and final investigation focused on the PAMs. Experimental temperature measurements were used to generate realistic two-dimensional CFX models of the thermal plumes emanating from the PAM electronics bays.

Figure 1. Schematic diagram showing the integration of thermal studies associated with NIF heat sources.

2. Light Fixture Evaluation

2.1 General Modeling Approach of a Fluorescent Light Fixture

The fluorescent light fixtures in the NIF laser bays generate significant amounts of waste heat that is dissipated primarily in the forms of radiation and convection heat transfer. While the HVAC system may be designed to remove the waste heat generated by the light fixtures, it may not be capable of adequately controlling the spatial variation in the facility's air temperature. It was the purpose of this study to understand how a fluorescent light fixture dissipates its heat and to what extent this influences the air temperature distribution in the NIF laser bay. This section summarizes the heat dissipation characteristics of a fluorescent light fixture as determined in a previous experimental study conducted by one of the authors [1]. In addition, the modeling approach used to determine the magnitude and extent of thermal disruption of the laser bay air by the light fixtures is discussed.

2.1.1 Heat Dissipation Characteristics of a Fluorescent Light Fixture

Bernardin et al. [1] conducted an extensive experimental study to determine the heat dissipation characteristics of a typical fluorescent light fixture. The results of that study for a two bulb light fixture are summarized in Figure 2. Of 81.6 W of power used by the light fixture, approximately 26.0 W (32%) is dissipated by convection from the plastic cover and metal housing, and the remaining 55.6 W (68%) is dissipated by radiation heat transfer. This distribution of the heat dissipation from a fluorescent light fixture is fairly consistent with that reported in the Marks' Standard Handbook for Mechanical Engineers [3].

Figure 2. Schematic diagram of the heat flow distribution from a fluorescent light fixture assembly with cover plate.

2.1.2 CFD Code

The numerical analysis of the thermal plume from the light fixture into the airflow involved simultaneously solving the continuity, Navier-Stokes, and energy equations. CFX 4.2 [2], a computational fluid dynamics computer code, was used to perform this task. CFX is a finite volume, implicit Navier-Stokes solver that uses a revised version of the mathematical algorithm Semi-Implicit Method for Pressure-Linked Equations, termed SIMPLEC [4]. Further information concerning the differencing schemes used to discretize the differential equations and the solution process can be found in [2]. All simulations were run on either a Silicon Graphics Indigo II (250 MHz IP22 processing unit and a 256 MB RAM) or a DEC-Alpha (500 MHz processor and a 512 MB RAM) workstation.

2.2 Light Fixture Suspended from a Beam

The first of three CFX fluorescent light fixture models in the NIF laser bay represented a light fixture suspended horizontally from a steel skeletal structural component such as the framework used to mount and align the various laser amplifier components. This type of light fixture would supply direct heating to the downward flowing cooling air as well as radiant heating of structural components in the near vicinity of the light. The purposes of this model were to determine the maximum recommended power rating of the lights as well as a 'sphere of influence' around the light fixture in which placement of critical beam alignment components should be avoided.

2.2.1 Model Geometry, Boundary Conditions, and Grid Discretization

The geometrical model for the numerical simulations is shown in Figure 3. A two-dimensional geometry was used to simplify the problem but still yield meaningful, worst-case results. The rectangular light fixture was modeled as a uniform heat flux surface with the heat flux selected to supply a representative convective heat flow into the HVAC airflow. Table 1 lists the types of fluorescent light fixtures being modeled along with the amount of heat dissipated by convection. Radiation heat transfer was not modeled in this study because the feature of interest was the convective thermal plume from the light fixture. Consequently, the amount of heat dissipated from the light fixture in the model represents only that portion of the total heat generated by the light fixture that is dissipated by convection heat transfer (40%). Since air is accurately modeled as a non-participating radiation medium, radiation heat transfer would not influence the thermal plume in this study. An analytical radiation heat transfer model was developed previously [1] and is used along with the CFX thermal plume information to determine minimum separation distances between lights and beam transport hardware.

Table 1. Temperature and heat transfer rate dissipation characteristics of beam mounted fluorescent light fixtures used in the CFX model.

Light Fixture	Total Power Dissipation (W) (10% Ballast)	Convection Heat Transfer (W) (40% of Total)	CFX Model Light Surface Heat Flux (W/m ²)
Two 32 W fluorescent bulbs	70.4	28.2	38.0
One 32 W fluorescent bulb	35.2	14.1	19.0

Note: All Dimensions in Meters,
Depth of Model = 1.219

Figure 3. Geometrical model and boundary conditions for the CFX light fixture analysis.

To represent the downward airflow from the vented ceiling ducts, a uniform inlet airflow profile was established on the top boundary of the model. Air at atmospheric pressure and 20.00°C was used as the cooling medium with an inlet velocity of 0.063 m/s (12.5 fpm).

The width of the model was based upon the 3.048 m (10 ft) center-to-center spacing of the majority of the light fixtures in the NIF laser bays, as specified in the NIF Beam Transport System Title II design drawings. Thus symmetry boundaries were placed on either side of the light fixture model. The extent of the model in the flow direction was made sufficiently long to prevent any downstream boundary conditions from communicating information back upstream. In this model, structural hardware was not included since the thermal plume was the feature of interest.

Other modeling conditions included steady-state buoyant and turbulent flow, as well as convection heat transfer. In addition, since the temperature and density variations were relatively small, the Boussinesq approximation was made which allowed the air to be modeled as an incompressible medium.

The grid generator for CFX 4.2 uses a multi-block scheme with a body-fitted grid structure. In this study, a geometric progression grid with 46,000 cells was used to accurately capture the boundary layers and flow structure around the light fixture.

2.2.2 Convergence Criteria

To determine the convergence and credibility of the CFX model, a three-step validation procedure of the numerical solution was employed. First and foremost, the convergence of a model onto a solution was determined with *residuals*. In the solution

algorithm of CFX, each dependent variable, y (i.e., temperature), is solved for in algebraic equations of the form

$$f(y) = F, \quad (1)$$

where F is a forcing function.

The residual for the dependent variable, y , is the error or difference between the left and right hand sides of Equation (1). The residuals for each dependent variable were determined at each cell and summed over all cells for a single iteration. A converged solution was obtained when the sum of the residuals for each dependent variable ceased to vary between consecutive iterations.

The final two tests of convergence were mass and energy balances of the model system. For the steady-state problems considered in this study, convergence was assumed when there was less than a 5% difference in the inlet and outlet flows of mass and energy.

These convergence tests were used in all three light fixture CFX models discussed in this report.

2.2.3 Results/Design Recommendations

Temperature contour and velocity vector plots for the domain surrounding a suspended light fixture are displayed in Figures 4 and 5 for light fixture powers of 70.4 W and 35.2 W, respectively.

The thermal impact of the light fixture waste heat on the surrounding air is clearly evident in Figure 4. The temperature contour and velocity vector plots of Figure 4 indicate that a small thermal plume is directed upward from the light fixture in opposition to the downward directed cooling air. The thermal plume is quickly dissipated within a meter of the light fixture, and the diffused heat is eventually carried away by the downward flowing cooling air. Within roughly a 1 m radius of the light fixture, the air temperature is above the 20.28°C limit desired for the ambient air temperature in the NIF laser bay. Consequently, all beam transport and optical alignment hardware, whose operating performance could be impacted by temperatures exceeding the designed NIF operating environment temperature field of $20.00 \pm 0.28^{\circ}\text{C}$, should not be placed within 1 m of the two 32 W bulb fluorescent light fixtures.

Figure 5 displays the temperature contour and velocity vector plots corresponding to a single bulb fluorescent light fixture (35.2 W). As expected, by reducing the power rating of the light fixture by half (as compared to the model in Figure 4), the extent of the thermal plume in excess of 20.28°C is reduced by half, to a radius of approximately 0.5 m around the light fixture.

Caution must be exercised when placing light fixture-to-hardware separation distance limits based upon the extent of thermal plumes. Recall that a fluorescent light fixture dissipates roughly 60% of its energy by radiation, a heat transfer mode not accounted for in the light fixture model used to produce Figures 4 and 5. In a previous NIF light fixture analysis by Bernardin et al. [1], in which minimum separation distances between light fixtures and beam transport tubes were desired, the radiation heat transfer, rather than the convective thermal plume, dictated the separation distance limits.

Figure 4. Numerical model (a) temperature contour and (b) velocity vector plots of the thermal plume from the suspended light fixture model with a total convective heat dissipation rate of 28.2 W, a maximum light surface temperature of 48°C, and a uniform inlet air velocity of 0.064 m/s.

Figure 5. Numerical model (a) temperature contour and (b) velocity vector plots of the thermal plume from the suspended light fixture model with a total convective heat dissipation rate of 14.1 W, a maximum light surface temperature of 32 °C, and a uniform inlet air velocity of 0.064 m/s.

Table 2 gives the minimum light fixture to beam transport/alignment hardware separation distances for two different light fixture power ratings. The thermal plume limits were derived from Figures 4 and 5 in the current study, while the analytical radiation model limits were taken from a previous study [1]. The light fixtures suspended in the NIF laser bay shine their light in a downward direction. Consequently, for hardware positioned above a light fixture, and hence shadowed from the majority of the light's radiation, the thermal plume dictates the minimum separation distance. For hardware positioned below a light fixture, and hence exposed to the majority of the light's radiation, the radiation heat transfer dictates the minimum separation distance. These two different separation distances are presented in Table 2 for the two different light fixtures.

Table 2. Minimum light fixture to beam transport/alignment hardware separation distances.

Light Fixture	CFX Thermal Plume Model Limit (m)	Analytical Radiation Model Limit [1] (m)	Recommended Design Limit (m) (Hardware Above Light Fixture)	Recommended Design Limit (m) (Hardware Below Light Fixture)
Two 32 W fluorescent bulbs	1.0	3.5	1.0	3.5
One 32 W fluorescent bulb	0.5	2.3	0.5	2.3

2.3 Light Fixture Mounted to a Wall, 2-D Model

The second CFX light fixture model was constructed to represent a light fixture mounted horizontally on a vertical wall in a concrete enclosure (i.e., transport spatial filter or periscope structures in the NIF laser bays) in which forced air circulation may be unpredictable. This type of light fixture would supply direct radiation and indirect convection heating to beam transport tubes passing over the top of the enclosure as well as to any structural or alignment components in the near vicinity of the light. It must be emphasized that this model is meant only as a design guide and not an exact representation of a particular structure. The purpose of this model was to determine the maximum light fixture power rating to prevent significant heating of beam transport tubes at the top of the enclosure.

2.3.1 Model Geometry, Boundary Conditions, and Grid Discretization

The geometrical model for the numerical simulations is shown in Figure 6. A two-dimensional geometry was used to simplify the problem but still yield meaningful, worst-case results. The rectangular light fixture was modeled as a heat flux surface with the value of the heat flux selected to supply both representative convection (40%) and radiation heat flows (60%) into the environment. The distribution of heat between convection and radiation modes was adjusted by manipulating the emissivity of the light surface. In addition, the light fixture was representative of a model that shines light in both the upward and downward directions. Table 3 lists the types of fluorescent light fixtures used in modeling, along with the amounts of heat dissipated by convection and radiation in the upward and downward directions. Radiation heat transfer was included in

Notes: All Dimensions
in Meters.
Depth of
Model = 1.219 m.

Figure 6. Geometrical model and boundary conditions for the two-dimensional enclosure light fixture analysis.

this model because of the close proximity of the light to the beam transport tubes (represented by the ceiling). Consequently, heating of the transport tubes resulted from a combination of direct radiant heating as well as indirect convective heating from the light fixture.

Table 3. Temperature and heat transfer rate dissipation characteristics of wall mounted fluorescent light fixtures used in the 2-D CFX model.

Light Fixture	Total Power Dissipation (W) (10% Ballast)	Convection Heat Transfer (W) (40% of Total)	Radiation Heat Transfer (W) (60% of Total)
Three 32 W fluorescent bulbs	106.0 (33% upward, 67% downward)	42.4	63.6
One 32 W fluorescent bulb	35.2 (uniformly distrubuted)	14.1	21.1

Air at atmospheric pressure and 20.00°C was used as the cooling medium. Since this model was constructed to represent a light fixture mounted in a concrete enclosure in which forced air circulation may be unpredictable, no forced cooling air was supplied at the model boundaries. Air circulation was developed purely by buoyancy forces.

The height of the model was chosen to be representative of the concrete enclosures in the NIF laser bays (i.e., transport spatial filter and periscope structures), as specified in the NIF Beam Transport System Title II design drawings. The location of the light was chosen purely as a representative position and to study how the thermal plume developed and traveled from the light fixture. It must be re-emphasized that this

model is meant only as a design guide and not an exact representation of a particular structure.

Other modeling conditions included steady-state buoyant and turbulent flow, as well as convection heat transfer. In addition, since the temperature and density variations were relatively small, the Boussinesq approximation was made which allowed the air to be modeled as an incompressible medium. A Monte Carlo radiation model was used with 2,500,000 photon histories calculated every 25th iteration of the CFX solver.

The grid generator for CFX 4.2 uses a multi-block scheme with a body-fitted grid structure. In this study, a geometric progression grid with 27,324 cells was used to accurately capture the boundary layers and flow structure around the light fixture and walls.

2.3.2 Results/Design Recommendations

Temperature contour and velocity vector plots for the domain surrounding a light fixture mounted on a vertical wall are displayed in Figures 7 and 8 for light fixture powers of 105.6 W (three bulbs) and 35.2 W (single bulb), respectively.

The thermal plume created by the waste heat from a three-bulb fluorescent light fixture (33% upward, 67% downward) is clearly evident in Figure 7. The temperature contour and velocity vector plots of Figure 7 indicate that a large thermal plume is directed upward from the light fixture along the insulated wall and is turned and directed horizontally by the ceiling structure. Heat is supplied to the thermal plume by convection heat transfer from the light surface as well as from portions of the wall and ceiling that are heated by radiation from the light. The radiant heating of the wall and floor beneath

the light fixture is also evident in the temperature contours of Figure 7. The air temperature within the thermal plume is well above the 20.28°C limit desired for the ambient air temperature in the NIF laser bay. The combination of radiant heating from the light and convective heating from the thermal plume raises the ceiling (beam transport tubes) temperature into the range of 21.0°C to 23.0°C. As reported in [1], beam transport tube surface temperatures within this range can create significant beam steering and distortion errors.

Figure 8 displays the temperature contour and velocity vector plots corresponding to a single bulb fluorescent light fixture (35.2 W) with its heat evenly distributed. As expected, by reducing the power rating of the light fixture by two-thirds (as compared to the model in Figure 7), the extent and strength of the thermal plume is significantly reduced. However, the ceiling, representing the underside of beam transport tubes, possesses surface temperatures in the range of 20.0°C to 21.5°C. As stated previously [1], beam transport tubes having surface temperature differentials (opposite facing surfaces) greater than 0.28°C can produce laser beam distortion. However, since the lengths of the transport tubes above the enclosures in which the wall mounted lights reside are relatively short, the amount of beam distortion may be negligible. In addition, the present model is conservative in that it does not account for three-dimensional dispersion of the light fixture heat, conduction dispersion of heat in the enclosure and transport tube walls, as well as forced convection cooling of the enclosure heat sources by ventilation systems (included in the next light fixture model). Consequently, it is believed that by reducing the single fluorescent light fixture power from the present

Figure 7. Numerical model (a) temperature contour and (b) velocity vector plots of the thermal plume from the wall mounted light fixture 2-D model with a combined total radiative and convective heat dissipation rate of 105.6 W, a maximum light surface temperature of 46.4 °C, and no forced convective airflow.

Figure 8. Numerical model (a) temperature contour and (b) velocity vector plots of the thermal plume from the wall mounted light fixture 2-D model with a combined total radiative and convective heat dissipation rate of 35.4 W, a maximum light surface temperature of 29.4 °C, and no forced convective airflow.

design of 105.6 W down to 35.2 W in the NIF laser bay enclosures, beam steering and distortion effects can be minimized to acceptable levels.

2.4 Light Fixtures Mounted to a Wall, 3-D Model

The third CFX light fixture model was constructed to represent light fixtures mounted in a realistic concrete enclosure representing the support structure beneath the transport spatial filter in the NIF laser bays. This three-dimensional model used a more rigorous approach than was employed with the previous 2-D model. In this model, the actual structure geometry and light fixture layout were included as well as forced air circulation provided by an HVAC air outlet duct located in the floor of the enclosure. As with the 2-D model, the light fixtures in the 3-D model supplied direct radiation and indirect convection heating to beam transport tubes passing over the top of the enclosure, as well as any structural or alignment components in the near vicinity of the lights (not included in the model for simplicity). The purpose of this model was to determine a *sphere of influence* around a light fixture in which placement of critical beam alignment components should be avoided and determine the maximum light fixture power rating to prevent significant heating of beam transport tubes at the top of the enclosure.

2.4.1 Model Geometry, Boundary Conditions, and Grid Discretization

The three-dimensional light fixture model is displayed in Figure 9. The model represents one-quarter of the transport spatial filter (TSF) enclosure and contains two symmetry boundaries to reflect this. The outer walls, floor, and ceiling were conservatively modeled as insulated boundaries, with an emissivity of 0.8 applied to the

Notes:

- * Back and Left Side are Symmetrical Boundaries
- * Top, Bottom, Right Side, and Front are Insulated
- * Walls with Emissivities = 0.8
- * Drawing not to scale
- * Front and Side Doorways are Pressure Boundaries
- * Light Fixtures Dissipate 105.6 W of Power Each, 40% Upward, 40% Downward, and 20% through the Sides
- * All Dimensions in Meters

Figure 9. Geometrical model and boundary conditions for the three-dimensional enclosure light fixture analysis.

floor and walls (concrete) and a value of 0.3 applied to the ceiling (steel). The ceiling in this model represents the underside of beam transport tubes and TSF assembly passing over the TSF enclosure.

Air at atmospheric pressure and 20.00°C was used as the cooling medium. The two doorways in the enclosure were modeled as atmospheric pressure boundaries, and a small outlet boundary, with a uniform downward velocity of 2.37 m/s, was positioned in the floor section of the model to represent an HVAC air return duct.

The three rectangular light fixtures were modeled as heat flux surfaces, the value of the heat flux was selected to supply both representative convection (40%) and radiation heat flows (60%) into the environment. The distribution of heat between convection and radiation modes was adjusted by manipulating the emissivity of the light surface. In addition, the light fixture was representative of a three-bulb, 105.6 W model that shines equal amounts of light in both the upward and downward directions. Figure 10 displays the heat distributions on the various sides of the lights for the two wall mounted and the single beam suspended fixtures. In both light fixture models, 80% (84.5 W) of the total heat dissipation was assumed to pass through the upper and lower surfaces, while 20% (21.1 W) was assumed to pass through the remaining sides. As with the two-dimensional wall mounted light fixture model discussed previously, radiation heat transfer was included in the three-dimensional model because of the close proximity of the light to the beam transport tubes (represented by the ceiling). Thus, heating of the transport tubes resulted from a combination of direct radiant heating as well as indirect convective heating from the light fixture.

(a)

(b)

Figure 10. Schematic diagram of the heat dissipation characteristics of the three bulb, 105.6 W, (a) wall mounted and (b) beam mounted fluorescent light fixtures used in the three-dimensional enclosure light fixture model.

Other modeling conditions included steady-state buoyant and turbulent flow, as well as convection heat transfer. In addition, since the temperature and density variations were relatively small, the Boussinesq approximation was made which allowed the air to be modeled as an incompressible medium. A Monte Carlo radiation model was used with 5,000,000 photon histories calculated every 25th iteration of the CFX solver.

The grid generator for CFX 4.2 uses a multi-block scheme with a body-fitted grid structure. In this study, a geometric progression grid with 306,600 cells was used to accurately capture the boundary layers and flow structure around the light fixtures and walls.

2.4.2 Results/Design Recommendations

Figure 11 displays temperature contour plots for four vertical planes at different locations in the three-dimensional light fixture model. Inspection of these temperature contours suggests that the buoyancy driven plumes drive heat upward towards the ceiling and that little waste heat is advected downward by the floor air duct. The temperature contour second from the right clearly shows the development of the thermal plume. Its characteristics are very similar to those previously predicted with the two-dimensional wall mounted light fixture model. The contour plots of Figure 11 also indicate that relatively large temperature gradients exist within the TSF enclosure with a significant portion of the air possessing temperatures above the upper design limit of 20.28°C. As mentioned earlier, temperature nonuniformities can have a detrimental impact on the optical support structures and alignment hardware that reside within the TSF enclosure. In addition, the convective plumes and radiation heating create undesirable temperatures

on the beam transport tube and TSF surfaces forming the ceiling of the TSF enclosure. The model predicts ceiling temperatures ranging from 20.0°C to 23.0°C, which obviously exceeds the upper temperature design limit.

While different light fixture power ratings were not investigated with the current model, it is believed that by reducing the single fluorescent light fixture power from the present design of 105.6 W down to 35.2 W in the NIF laser bay enclosures, the spatial distribution of air temperatures can be kept below the upper design limit of 20.28°C. It is further recommended that motion detectors, connected to the light fixture power supplies within the laser bay enclosures, be employed to shut off the lights when maintenance/operations personnel are not present. Finally, the minimum separation distances between light fixtures and critical beam transport and optical alignment components, as listed in Table 2, should be adhered to.

2.5 Summary of Light Fixture Design Recommendations

Three different CFD light fixture models were developed to investigate the detrimental impact of light fixture waste heat in raising the temperatures of the laser bay air, beam transport structures, and optical alignment hardware. The models yielded design guidelines such as the maximum recommended light fixture power ratings as well as minimum separation distances between the light fixtures and critical beam transport and alignment components.

In summary, the design modifications/recommendations for the fluorescent light fixtures in the NIF laser bay are as follows:

Figure 11. Numerical model temperature contour plots at four different vertical planes in the wall mounted light fixture 3-D model with a total heat dissipation rate of 316.8 W (for three lights) and a maximum light surface temperature of 47°C.

The power of each light fixture in the current design should be reduced to approximately 34 W, corresponding to a single bulb fixture. This will require installing either single bulb fixtures or multiple bulb fixtures with the understanding that only one bulb will be placed in each fixture.

- Motion detectors, connected to the light fixture power supplies inside the NIF laser bay concrete enclosures (support structures), should be employed to shut off the lights inside these enclosures when maintenance/operations personnel are not present.
- The minimum separation distances between light fixtures and critical beam transport and optical alignment components should be adhered to. Table 4 lists these minimum separation distances for both single and double bulb light fixtures. Also note that the recommended separation distance depends on whether or not the hardware is in direct illumination by the light fixture. If these minimum separation distances cannot be met for certain situations, then it is recommended that the hardware within close proximity to the light fixture be insulated with 1.27-cm-thick foam-board insulation with a shiny aluminum foil outer covering.

Table 4, Minimum light-fixture to beam transport/alignment hardware separation distances.

Light Fixture	CFX Thermal Plume Model Limit (m)	Analytical Radiation Model Limit [1] (m)	Recommended Separation Distance (m) for Hardware Exposed to Direct Lighting from the Light Fixture	Recommended Separation Distance (m) for Hardware Shadowed from Light Fixture
Two 32 W fluorescent bulbs	1.0	3.5	1.0	3.5
One 32 W fluorescent bulb	0.5	2.3	0.5	2.3

As a final consideration, while reducing the light fixture power in the NIF laser bay may benefit the NIF operating performance, it may lead to maintenance or safety problems. It is further recommended that the NIF laser bay lighting engineers ensure that the final optimized lighting plan satisfies OSHA requirements for minimal safety lighting as well provides sufficient lighting for operation and maintenance personnel.

3. Electronics Rack Evaluation

This study focuses on the individual heat generating electronics racks located throughout the NIF laser bay. The electronics racks are storage cabinets which house electronic equipment such as controllers, power supplies, instrumentation packages, and various other components which are responsible for heat generation inside the electronics racks. The electronics racks are located beneath the laser beam tubes in the NIF and are known to contribute unsatisfactory amounts of waste heat to their surroundings if not properly cooled. The purpose of this study is to understand the magnitude and extent of thermal plumes emanating from the outer surfaces of the electronics rack under various heat dissipation rates.

3.1 Heat Dissipation Characteristics of an Electronics Rack

An experiment was conducted at Lawrence Livermore National Laboratory to determine the heat dissipation characteristics of an electronics rack with a total operating power of approximately 1000 W. Figure 12 displays the side-by-side electronics rack setup used in this study. Mock-up electronics boxes were constructed and placed inside Rack 1 to assist in creating obstacles for the circulating cooling airflow. A total of

1000 W of power was supplied to Rack 1 by five 200 W light bulbs. Most of the generated heat was removed from the system by an air-to-water heat exchanger which was located near the bottom of the electronics rack. The air-to-water heat exchanger used cooled water, which was supplied by a remote water chiller, to remove heat from warm circulating air in the electronics rack.

Thermocouples were placed at various locations on and around the experimental electronics racks as indicated in Figure 12. Table 5 summarizes the average temperatures at the various thermocouple locations.

Table 5. Electronics rack experiment using a 1000 W power input and a water coolant mass flow rate of 0.063 kg/s per rack.

Thermocouple #	Measurement	Temperature (°C)
1	Lab Air	22.50
2	Inlet Water	12.81
3	Outlet Water	16.55
4	Inlet Air	22.56
5	Outlet Air	16.67
6	Rack 1 Top	23.61
7	Rack 2 Top	23.50
8	Rack 1 Side	20.10
9	Rack 2 Side	26.78

Table 5 indicates that the temperatures of the electronics rack tops are slightly higher than lab air temperature. Consequently, a small amount of waste heat was being dissipated to the air surrounding the electronics racks. The heat removal rate of the heat exchanger can be computed analytically using an energy balance on the coolant water

$$q_{HE} = m c_p (T_{out} - T_{in}), \quad (2)$$

where m is the water mass flow rate through the heat exchanger, c_p is the specific heat of water, and T_{out} and T_{in} are the heat exchanger's outlet and inlet temperatures, respectively. In applying Equation (2) to the experimental study (water mass flow rate of 0.063 kg/s), the energy balance on the heat exchanger indicated that the heat exchanger removed approximately 987 W (± 250 W due to the $\pm 0.5^\circ\text{C}$ uncertainty in the thermocouple measurements). Consequently, the heat exchanger appears to have removed the majority of the 1000 W waste heat from the electronics rack experiment.

To further investigate the waste heat dissipation from the electronics rack to the surroundings, the present numerical study was conducted using CFX. The following section discusses the properties of the CFX models used in the study.

3.2 Model Geometry, Boundary Conditions, and Grid Discretization

The geometry used in the analysis of the electronics rack is displayed in Figure 13. The two-dimensional model, also displayed in Figure 13, was employed to represent a typical electronics rack in the NIF. Although the geometry of the electronics racks is simplified, it still provides an accurate representation for examining heat transfer and airflow characteristics. Simplifications and assumptions, such as those made in this model, allow for decreased computational error and shorter computer run times.

Note: Thermocouples 8 & 9 are located beneath the insulation

Figure 12. Schematic diagram of the electronics racks experimental apparatus.

Figure 13. Geometrical Model for the CFX electronics rack analyses.

Figure 14 displays all boundary conditions that were employed on the electronics rack model. First, the model size was reduced by taking advantage of the centerline symmetry plane. Next, the electronics rack was modeled as a heat source with waste heat escaping through its outermost surface boundary. In most all cases, this was accomplished by using a constant uniform heat flux on the external surface of the electronics rack. Heat flux was determined by multiplying the total power input by the heat exchanger heat removal inefficiency (%) and then dividing the product by the total external area. For example, if an electronics rack, with an input power of 1000 W and an external area of 3 m², has a 10% heat exchanger inefficiency, then the heat flux is 33.33 W/m². A total of 100 W of power will escape to the surrounding air. A separate case employed a linear temperature profile, ranging between 20.0°C and 26.0°C, along the side wall of the electronics rack. The top wall, in this case, possessed a constant uniform temperature of 26.0°C. Air at 20.0°C was used as a cooling fluid. An inlet boundary condition was placed at the top of the model to imitate forced downward airflow from HVAC vents above. Inlet velocities ranged from 0.000 m/s to 0.064 m/s and were directed downward toward the electronics rack. Finally, pressure boundaries, at atmospheric pressure, were logically placed on other areas to simulate the effect of a large open room.

The model was meshed by the grid generating subprogram in CFX - Meshbuild. A total of 84,357 grid cells were modified, using geometric progression, to allow for an increased number of grid cells to be placed in regions where large temperature and velocity gradients existed.

3.3 Convergence and Analytical Verification

A number of steps were taken to determine convergence and credibility of the results produced in CFX. The same convergence tests used for the light fixture analyses, and described in Section 2.2.2, were used in the electronics rack models. As discussed previously, an energy balance of 5% to 10% was deemed acceptable. However, in the electronics rack models employing a 0.032 m/s forced airflow, the energy balance is not acceptable. These cases were provided in this report so that a general idea could be gained for cases with forced flow velocities between 0.000 m/s and 0.064 m/s (see Table A.1 in the Appendix).

Basic analytical calculations were performed to assess the validity of results obtained in this study. First of all, the relative strengths of the buoyancy flows, driven by the warm surfaces of the electronics rack and the opposing HVAC airflow, were compared with the ratio of the Grashof number, Gr_L ,

$$Gr_L = \frac{g\beta (T_s - T_{Surr})L^3}{\nu^2}, \quad (3)$$

where

$$\beta = \frac{2}{(T_s + T_{Surr})}, \quad (4)$$

and Reynolds number squared, Re_L^2 (Gr_L / Re_L^2)

$$Re_L = \frac{VL}{\nu}. \quad (5)$$

Figure 14. Boundary conditions for the CFX electronics rack analyses.

Free convection dominates over forced convection if $Gr_L \gg Re_L^2$ and vice versa if $Re_L^2 \gg Gr_L$. In all cases studied in this investigation, the buoyancy flow was significantly stronger than the opposing forced airflow. This is evident in the velocity vector plots which are displayed in Section 3.4.

To determine the presence of turbulence in the buoyant airflow, the magnitude of the Raleigh number was calculated and compared against the experimentally determined *transition to turbulence* value of 10^9 [5].

$$Ra_L = \frac{g\beta (T_s - T_{Surr})L^3}{\nu \alpha} \approx 10^9. \quad (6)$$

The Raleigh number was calculated, for example, in the 100 W output case, with no forced flow. In this case, $L = 2.2098$ m, $T_s = 35^\circ\text{C}$, $T_{Surr} = 20^\circ\text{C}$, $\beta = .003328 \text{ K}^{-1}$, $g = 9.81 \text{ m/s}^2$, $\nu = 1.527\text{E-}5 \text{ m}^2/\text{s}$, and $\alpha = 2.158\text{E-}5 \text{ m}^2/\text{s}$. The final calculation revealed the Raleigh number, $Ra_L = 1.604\text{E}10$, to be greater than 10^9 . Therefore, the airflow in the electronics rack model proved to be turbulent.

To confirm electronics rack surface temperatures predicted by CFX, the following convective heat flux, q'' , equation was used:

$$q'' = h(T_s - T_{Surr}). \quad (7)$$

To solve for the average surface temperature, T_s , in Equation (7), the free convection heat transfer coefficient, h , needed to be determined. This was done by solving the following empirical Nusselt correlation, recommended by [6], for h :

$$Nu_L = \frac{hL}{k} = \left\{ 825 + \frac{387 Ra_L^{1/6}}{\left(1 + (.492 / Pr)^{1/6}\right)^{8/27}} \right\}^2. \quad (8)$$

The Rayleigh number was again found using Equation (6). This required an iterative procedure since T_s , needed to solve Ra_L , was not known a priori. Equations (6), (7), and (8) were simultaneously solved in Microsoft® Excel. Average surface temperatures predicted using these equations closely corresponded (>2% error) to those predicted with the CFX model for cases with no downward flow.

3.4 Results

Twelve unique electronics rack cases were examined. Each of these cases were run with a downward inlet velocity of either 0.000, 0.032, or 0.064 m/s. Most models of the electronics racks carried an input power of 50, 100, 250, 500, or 1000 W, which was uniformly distributed on the outer surface. An additional case used a linear surface temperature profile boundary condition, between 20°C and 26°C, along the vertical walls of the electronics rack. In this case, the top of the rack possessed a uniform temperature of 26°C. Table A.1 in the Appendix summarizes the boundary conditions and parameters used in the twelve models.

To keep the discussion of results manageable and focused, four models which span the various boundary conditions were selected and are discussed in detail in the following sections. Temperature contour and velocity vector plots of the remaining models are deferred to the Appendix.

3.4.1 Linear Temperature Profile Boundary with No Downward Airflow

Figure 15 is a temperature contour plot for an electronics rack with a linear temperature profile distributed along the right-side wall. The temperature profile varies linearly between 20°C and 26°C in the vertical direction from the bottom of the electronics rack. The effect of the thermal plume is evident in Figure 15. The thermal plume in this case possesses a maximum temperature of 26°C near the surface of the electronics rack and gradually decreases as it rises and dissipates heat to the cooler surrounding air. At approximately 2.65 m above the electronics rack, the air temperature drops to just below the specified design limit of 20.28°C. It is therefore recommended that all optical alignment and beam transport equipment be located at a minimum distance of 2.65 m above the top surface of an electronics rack with a heat dissipation characteristic of that displayed in Figure 15. The widths of the thermal plumes emanating from the sides of the rack are quite small and, consequently, hardware can be positioned to within 0.1 m of the rack sides.

Figure 15. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack with a linear temperature profile ranging from 20 to 26 °C, and no forced external airflow.

3.4.2 Heat Flux Boundary with No Forced Airflow

Figures 16 and 17 show temperature contours and velocity vectors for thermal plumes rising off of electronics racks that are dissipating 100 W and 1000 W of waste heat, respectively. In both cases, no forced airflow is introduced from the HVAC inlets located above the electronics rack. Consequently, in each of these cases, the thermal plume rises unobstructed by natural convection.

The temperature contour plot for the 100 W case in Figure 16 exhibits an unacceptable amount of waste heat leaving the electronics rack. The majority of the thermal plume within 4.85 m above the top of the rack possesses temperatures well above the specified upper design limit of 20.28°C. Consequently, for an electronics rack dissipating roughly 100 W to the laser bay air, a minimum separation distance of 4.85 m must be maintained between the upper surface of the rack and critical beam transport hardware.

Figure 17 displays the 1000 W electronics rack case with no downward flow. The magnitude and extent of the thermal plume for this case are significantly greater than those observed previously for the 100 W case. The thermal plume air temperatures range from 135°C on the top surface of the electronics rack to 22.5°C at the top of the CFX model. For this case, the minimum separation distance beyond the top of the rack for beam transport hardware is well beyond the 4.85 m available in the model. This thermal disturbance to the laser bay air is obviously unacceptable and indicates that proper attention must be given to ensure that electronics racks are properly cooled.

Figure 16. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 100 W, a maximum skin temperature of 39 °C, and no forced external airflow.

Figure 17. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack with a leakage power of 1000 W, a maximum skin temperature of 135 °C, and no forced external airflow.

3.4.3 Heat Flux Boundary with Forced Airflow

Figure 18 displays the thermal plume behavior for the 100 W waste heat electronics rack with the addition of a downward directed airflow of 0.064 m/s at the upper model boundary. The thermal plume extends roughly 4.4 m beyond the upper rack surface, where it encounters the downward directed inlet air at the uppermost model boundary. At this point, the thermal plume is diverted and directed horizontally to the outer pressure boundary. A comparison of the temperature contour and velocity vector plots of Figures 16 and 18 reveals that the downward directed air is not very effective in dispersing the thermal plume. In fact, the thermal plume is diverted only because it has reached the inlet boundary where the downward air velocity is specified. As stated previously, analytical calculations of the Grashoff and Reynolds numbers (Equations 3 and 5) predicted this behavior for the modeling conditions investigated in this study.

The results from these models indicate that a heat exchanger is essential for properly cooling an electronics rack. In addition, depending upon the amount of waste heat emanating from the electronics rack, a minimum separation distance must be maintained between the upper surface of the rack and critical beam transport hardware, as detailed in Table 6.

Figure 18. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 100 W, a maximum skin temperature of 39 °C, and 0.064 m/s forced external airflow.

Table 6. Minimum electronics rack to beam transport/alignment hardware separation distances.

Electronics Rack Model	CFX Thermal Plume Model Limit (m)	Recommended Design Limit (m) (Hardware Above Electronics Rack)	Recommended Design Limit (m) (Hardware next to Front, Sides & Back)
Linear Temperature Profile	2.65	2.65	0.10
100 W, No Force Airflow	≈ 4.85	4.85	0.25
100 W, 0.064m/s Airflow	4.50	4.85	0.20
1000 W, No Forced Airflow	Beyond Model Upper Boundary	$\gg 4.85$	0.35

The minimum electronics rack to beam transport/alignment hardware separation distances for all models investigated in this study are given in Table A.1 in the Appendix. In addition, temperature contour and velocity vector plots for the eight remaining models not discussed in this section of the report are contained in the Appendix.

3.5 Summary of Electronics Rack Design Recommendations

Twelve CFD models were created to study the thermal pluming effects caused by waste heat dissipation from the electronics racks. The models provided information regarding the magnitude and extent of the thermal plumes emanating from the electronics rack surfaces. The results of the electronics rack experiment and the CFX modeling lead to the following design recommendations:

- An air-to-water heat exchanger will be required for all electronics racks with a total operating power between 25 and 1000 W.

- The power distribution of the various electronics racks in the NIF laser bay is shown schematically in Figure 19. It is seen that individual rack power generation rates vary from 0 to 5000 W. Consequently, the cooling capacities for an electronics rack must be consistent with its particular power output (i.e., if an electronics rack is producing only 50 W of waste heat, then the heat exchanger should be expected to remove all or most of the 50 W and nothing additionally). Overcooling of the electronics racks must be avoided to prevent undesired temperature gradients.
- Any electronics rack with an operating power above 1000 W may need multiple heat exchangers.
- Further experimental testing should be performed in order to predict the thermal properties of electronics racks with heat generation rates other than 1000 W.
- The minimum separation distances between the electronics rack and critical beam transport and optical alignment components, given in Tables 6 and A.1, should be adhered to.

Figure 19. Electronics rack power distribution layout in the NIF laser bay.

4. Pre-Amplifier Module (PAM) Evaluation

4.1 Heat Dissipation Characteristics of a PAM

The Pre-Amplifier Module (PAM) is a multi-functional, highly integrated two-stage laser amplifier package. Each PAM unit contains an optics bay, which is housed in an optical support structure (OSS). An electronics bay (e-bay) is placed on top of the OSS. Figure 20 illustrates the major assemblies in a PAM. The e-bay is made up of an electronic chassis with rack mounted components and is covered by aluminum sheets. The electronic components in the e-bay collectively produce 628 W of waste heat, according to PAM prototype power consumption measurements.

The waste heat generated in the e-bay must be removed. Forced airflow at a rate of $0.061 \text{ m}^3/\text{s}$ ($130 \text{ ft}^3/\text{min}$) through the e-bay cools the electronic components and chassis. The airflow is provided by inlet and outlet fans, which are located at each end of the e-bay. Before the warm air exits the e-bay, it is passed through an air-water heat exchanger, which uses about 1 gal/min of actively controlled flow of water at 7°C (45°F) to cool the air back to the facility nominal air temperature of 20.0°C .

In this thermal study, the goal is to determine the influence of the waste heat generation in the e-bays on the temperature of the air above the PAMs and inside the OSS. It is assumed (and was supported by prototype experiments) that the air-water heat exchanger effectively cools the exit forced air from the e-bay before it returns to the facility. The focus is then on the amount of heat generated in the e-bay that is not removed by the forced airflow. That amount of heat will be transmitted through the sheet-metal cover of the e-bay to warm the air above.

Figure 20. Major assemblies in a PAM. The sheet metal covering the electronics bay and the front cover of the optical support structure are not shown.

In the following sections, a description of two computational models used for these analyses is given. These models separately address the thermal effects of the PAM waste heat generation on the facility (laser-bay) and OSS air temperatures.

4.2 Model Geometry, Boundary Conditions, and Grid Discretization

Two PAM units are typically stacked one on top of another in the pre-amplifier support structure. Figure 21 depicts schematically such an arrangement, showing the e-bay and OSS of each PAM. Outlined in red are the two main computational models used in this study. Model A is used to analyze the effect of the e-bay waste heat generation on the air above the upper PAM unit, whereas Model B is used to study the effects on the air in the OSS.

In both Models A and B, the e-bay (3.42 m long, 0.182 m high, and 0.710 m wide) is idealized as consisting of all solid volume at the bottom and free air space at the top. The solid volume at the bottom represents the combination of all the components in the e-bay which act as flow obstacles. It is assumed that the solid obstacle volume occupies 80% of the total e-bay volume, leaving 20% free volume for the forced airflow to go through. This assumption leads to an average air velocity of 2.4 m/s (based on a volumetric flow rate of $0.061 \text{ m}^3/\text{s}$ divided by the idealized e-bay free-space cross-sectional area of 0.0258 m^2), which is consistent with prototype air velocity measurements in various locations within the e-bay.

Two-dimensional, steady-state CFX calculations were carried out for Models A and B. The ambient air, forced flow inlet, and enthalpy reference temperatures were all set to 20.0°C .

Figure 21. Schematic drawing of two PAM units depicting Models A and B used in thermal analysis studies.
 Note: Vertical dimensions not to scale.

In the following paragraphs, the geometry, boundary conditions, and spatial discretization for the two models are described.

Figure 22 shows the overall dimensions, material distribution, and boundary conditions used in Model A, which consists of the upper e-bay and the laser bay air above it. Volume occupied by a 0.635-cm-thick foam-board insulation on top of the sheet metal is included so that cases with and without insulating the top of the e-bay could be studied. Note that the vertical scales for the sheet metal and e-bay air are expanded. The horizontal dimension, which represents the length of the e-bay, is 3.42 m. The e-bay free-space height is 3.6 cm, and the sheet metal is 0.229 cm thick. The vertical dimension of 3 m for the outside (i.e., laser bay) air was judged to be high enough such that the constant pressure boundary condition at the top boundary does not affect the development of the thermal plume caused by the heated sheet metal.

In addition to the top boundary, the left and right boundaries of the outside air volume are also under constant pressure boundary condition, which will allow air to flow in or out depending on the fluid dynamics developed inside the computational domain.

For the e-bay air volume, the left boundary is specified with a constant mass inflow (74 g/s of air at 20.0°C) boundary condition that corresponds to 0.061 m³/s (130 ft³/min). On the right boundary, the mass outflow (or alternatively constant pressure) boundary condition is specified. On the bottom boundary of the e-bay air volume, a constant heat flux of 258.6 W/m² (628 W divided by the bottom wall area of 2.428 m²) is specified as this boundary also represents the top wall of all the e-bay components (which produce the waste heat) lumped together, which are not modeled directly here.

Figure 22. Overall dimensions, material distribution, and boundary conditions used in Model A, which consists of the upper e-bay and the laser bay air above it. Note: Vertical dimensions not to scale.

For spatial discretization in Model A, 100 uniformly spaced cells are used in the horizontal direction. In the vertical direction, 20 cells are used for the e-bay air, 3 for the sheet metal, 8 for the insulation (if exists), and 95 for the outside air. The total number of cells is 12,600. The cell sizes in the vertical direction are chosen such that more cells bunch up near solid walls to better resolve boundary layers.

Model B consists of the lower e-bay and the upper OSS, separated by an air space in between. Figure 23 illustrates the overall dimensions, material distribution, and boundary conditions used in this model. The e-bay, sheet metal, insulation material (if exists), and overall horizontal dimensions are the same as in Model A. The OSS is 0.787 m high and has a steel bottom wall that is 0.318 cm thick. The OSS is modeled as a closed volume in which airflow is not allowed in or out. The air space between the OSS and e-bay is 0.25 m high.

Boundary conditions for the e-bay air region are the same as in Model A. Similarly, the left and right boundaries of the outside air are under a constant pressure boundary condition. For the OSS, the top wall is specified with an adiabatic boundary condition. The left and right walls of the OSS are specified with a heat flux boundary condition that models cooling of the walls by natural convection of the outside (laser-bay) air. This boundary condition is derived as follows:

$$q_w'' = k \frac{\partial T_w}{\partial x} = h(T_\infty - T_w), \quad (9)$$

$$k \frac{\partial T_w}{\partial x} + hT_w = hT_\infty, \quad (10)$$

Figure 23. Overall dimensions, material distribution, and boundary conditions used in Model B, which consists of the lower e-bay and the upper OSS separated by an air space. Note: Vertical dimensions not to scale.

where T_w and q_w'' are the temperature and heat flux at the wall, k is the thermal conductivity of air in the OSS, T_∞ is the ambient air temperature (20.0°C), and h is the heat transfer coefficient that represents the heat removal capability of the ambient air via natural convection at the vertical OSS walls. The value of h chosen for the natural convection boundary conditions is 1.428 W/m²-K, which is determined from the following correlation developed by Churchill and Chu [6] based on a temperature difference (ΔT) of 1°C and the OSS height (L) of 0.787 m:

$$Nu_L \equiv \frac{hL}{k} = 0.68 + 0.67Ra_L^{1/4} \left[1 + (0.492 / Pr)^{9/16} \right]^{4/9}, \quad (11)$$

where

$$Ra_L \equiv \frac{g\beta\Delta TL^3}{\alpha\nu}. \quad (12)$$

Note that this correlation is applicable to both laminar and turbulent natural convection on vertical isothermal surfaces. The ΔT value of 1°C assumed above is verified to be roughly the calculated temperature difference between the vertical OSS boundaries and the ambient air (see Section 4.3).

The overall horizontal spatial discretization, as well as the discretization of the e-bay air region, sheet metal, and insulation in Model B, are the same as in Model A. The air space between the e-bay and OSS is meshed with 80 computational cells vertically while the OSS is meshed with 65 cells. Again, the distribution of cells is such that they converge toward solid boundaries to better resolve momentum and thermal boundary layers. The total number of cells used in Model B is 17,900.

In both Models A and B, conducting solid models are used to invoke conjugate heat transfer with the adjacent fluid (air) regions. Modeled conducting solid materials include the aluminum sheet metal, insulation material, and OSS steel wall. The thermal properties of these conducting solids are summarized in Table 7.

Table 7. Physical properties of conducting solids specified in CFX calculations.

Conducting Solid	Thermal Conductivity (W/m/°C)	Specific Heat Capacity (J/kg/°C)	Density (kg/m ³)
Sheet Metal (Aluminum)	237	905	2707
Steel	43	473	7801
Foam-board Insulation	0.01	1000	790

The calculations for Models A and B were performed both with and without radiation modeling. Radiation heat transfer was modeled in the CFX calculations with the Shah discrete transfer method, using twelve rays and one radiation call per five fluid and heat flow iterations. Surface emissivities for all inflow, outflow, and pressure boundaries are set to unity to represent a non-reflective radiation boundary condition. The other boundaries (solid surfaces) have emissivity values listed in Table 8.

Table 8. Emissivities of various solid surfaces specified in CFX calculations.

Material	Aluminum Sheet Metal	Steel	Foiled Foam-Board Insulation	E-Bay Solid Components
Surface Emissvity	0.09	0.66	0.07	0.5

All four boundaries of the OSS are assumed to be steel surfaces with an emissivity of 0.66. The emissivity value of 0.50 for the e-bay solid components is an approximation for a fictitious surface of the idealized solid volume that lumps all the flow obstacles together.

All of the calculations invoked the standard k - ϵ turbulence model. The fluid flow was modeled as incompressible, and buoyancy was modeled with the Boussinesq approximation.

4.3 Results

In all of the calculations, convergence to steady-state solutions was considered achieved when the residuals of all the nonlinear outer iteration equations decreased by at least three orders of magnitude between the second and last iterations. The nonlinear equations include two momentum, a pressure (mass), and enthalpy equations, as well as transport equations for k and ϵ . Because energy balance is especially important for the current problem, the enthalpy conservation equation was iterated four times in each outer iteration loop. Convergence of this equation was aided with false time stepping at a step size of 0.1 s. Global energy balance check for all calculations indicated that the energy was conserved to within 0.1%.

It was found that using double-precision for the current calculations was essential in obtaining well converged solution for the enthalpy equation within a reasonable number of iterations (about 2000). This is because even small temperature differences (e.g., 0.1 K in the absolute temperature scale, which is used by CFX computations, of about 300 K) are significant for the current problem, and the temperature solutions in

most regions are approximately the same (293.15 K or 20°C). Although the double-precision option needed about 50% more memory than single-precision in these calculations, the more accurate computations led to shorter CPU times and more straightforward convergence strategies.

Most calculations converged to acceptable steady-state solutions within 10 hours of CPU time. In the following sections, calculation results for Models A and B will be discussed separately.

4.3.1 Model A Results

The purpose of the Model A calculations is to assess the thermal plume above the upper e-bay that develops as a result of the waste heat generated by the electronic components. Calculations are presented without radiation modeling first, then those with radiation modeling. The effects of insulating the top of the sheet metal are also discussed.

Figure 24 shows the calculated temperature contours and velocity vectors when radiation modeling was not included for both the cases with and without insulation. Note that the temperature scale was chosen with a maximum at 20.28°C and minimum at 20°C. Therefore, regions with temperatures at or above 20.28°C all appear red on the contour plots. A thermal plume above the e-bay is clearly shown. If the plume is defined by the 20.23°C temperature contour line, then the plume height is 0.94 m for the case without insulation and is 0.39 m for the case with insulation. The thermal plume is caused by the amount of waste heat that is not carried away by the forced flow inside the e-bay. The

Figure 24. Calculated temperature contours and velocity vectors for Model A
(a) without insulation (base case) and (b) with insulation.

fractional amount of waste heat escaping the e-bay through the sheet metal is 2.3% for the case without insulation and 0.7% for the case with insulation.

Figure 24 also shows the flow patterns. The warm rising air that forms the thermal plume above the sheet metal is replenished by cooler air flowing into the computational domain from both the left and right sides. The maximum velocity magnitude within the plume is 11 cm/s for the case with no insulation and 10 cm/s for the case with insulation. Velocity vectors within the e-bay are not shown in Figure 24, because the e-bay height is too small compared to the air above it and because the vectors inside the e-bay would overwhelm those defining the thermal plume. The calculated velocity in the e-bay exhibits a typical profile for a turbulent forced channel flow, with a maximum velocity of 2.78 m/s at the center for all cases (i.e., Models A and B, with and without insulation and radiation modeling).

Other temperature results show that the sheet metal reaches a maximum of 24°C for the case without insulation and 25°C for the case with insulation. In both cases, the maximum temperature at the bottom wall, which gives off the waste heat, is 41°C, and the forced air flow exits the e-bay with a temperature profile of 38°C at the bottom and 25°C the top.

Calculations were performed to assess the effect of a higher waste heat generation, in which the power was increased by 10% from 628 W to 691 W. Figure 25 shows the resulting thermal plume for this higher-power case, together with the nominal power case with no insulation. It can be seen that the thermal plume is higher by about 40% (1.3 m compared to 0.94 m).

Figure 26 shows the temperature contours and velocity vectors with radiation modeling included in the calculations. It can be seen that the thermal plume is weaker than in the above case in which radiation modeling was not included. This is because radiation from the top of the sheet metal (or insulation) to the constant-pressure boundaries (which were assumed to be black-body heat sinks with an emissivity of unity) provides an additional cooling mechanism. The plume height is now 0.38 m for the case without insulation and is 0.21 m for the case with insulation. The percentages of the total power escaping the e-bay through the sheet metal for the cases without and with insulation are 3.3% and 0.9%, respectively. The maximum velocity within the plume is 6 cm/s for the case without insulation and is 4 cm/s for the case with insulation. The sheet metal maximum temperature is about the same as in the case without radiation modeling (i.e., 24°C without insulation and 25°C with insulation). The maximum temperature at the bottom wall is 40°C, and the maximum temperature of the forced airflow at the exit is 37°C, for both cases with and without insulation. Table 9 lists the important results for Model A.

Table 9. Summary of results for Model A.

	No Radiation Modeling		With Radiation Modeling	
	No Insulation	With Insulation	No Insulation	With Insulation
Plume Height (m)	0.94	0.39	0.38	0.21
% Total Power Through Sheet Metal	2.3%	0.7%	3.3%	0.9%

(a)

(b)

Figure 25. Calculated temperature contours and velocity vectors for Model A, comparing two waste heat levels of (a) 628 W (base case) and (b) 691 W.

1

2

3

Figure 26. Calculated temperature contours and velocity vectors for Model A, with radiation modeling
(a) without insulation and (b) with insulation.

4.3.2 Model B Results

With Model B, the effects of the waste heat generated within the lower e-bay on the air temperature inside the OSS above is investigated. As in Model A, calculation results for cases with and without radiation modeling, and with and without insulation on top of the sheet metal, will be presented.

Figure 27 shows the calculated temperature contours and velocity vectors when radiation modeling was not included for both the cases with and without insulation. Note that the scale for the temperature contours has a maximum of 22°C. As in Model A, a thermal plume develops in the air above the e-bay, which is caused by the warm sheet metal. In the current case, however, the plume is deflected horizontally as it hits the bottom wall of the OSS. As a result, the flow pattern in the air space between the lower e-bay and the upper OSS is such that warm air leaves the left and right boundaries at the top while cool air is drawn in at the bottom across the same boundaries. The fractional amount of the total waste heat that escapes the e-bay through the sheet metal is 1.6% for the case without insulation and 0.6% for the case with insulation.

As can be seen in Figure 27, two convection loops occur inside the OSS, with warm air rising near the middle, deflecting sideways at the top wall, then flowing downward along the two cooled side walls. The average temperature of the air in the OSS is 21.3°C without insulation and 20.7°C with insulation. About 1 W and 0.54 W, respectively, of heat enters the OSS from the bottom for the cases with no insulation and with insulation.

Other results, such as the maximum temperatures of the sheet metal, of the bottom wall giving off the waste heat, and of the exit forced flow of air, are very similar to those in Model A.

Figure 28 shows the temperature contours and velocity vectors with radiation modeling included in the calculations. Note the change in the maximum temperature on the contour color scale to 23°C. A general observation is that with radiation modeling included, the air within the OSS becomes slightly warmer than in the case with no radiation modeling. This is because the top and side walls (especially the top wall, which was specified with an adiabatic boundary condition), with an emissivity value of 0.66, absorb as well as reflect energy back into the computational domain. Similar flow patterns develop both inside the OSS and in the air space between the e-bay and OSS, as in Model A. However, the average temperature of the air inside the OSS in this model is 22.1°C without insulation and 21.7°C with insulation. The percentages of the total power escaping the e-bay through the sheet metal for the cases without and with insulation are 3.3% and 0.9%, respectively. Table 10 lists the major results for Model B.

Table 10. Summary of results for Model B.

	No Radiation Modeling		With Radiation Modeling	
	No Insulation	With Insulation	No Insulation	With Insulation
OSS Air Temperature Rise (°C)	1.3	0.7	2.1	1.7
% Total Power Through Sheet Metal	1.6%	0.6%	1.8%	0.7%

Figure 27. Calculated temperature contours and velocity vectors for Model B
(a) without insulation and (b) with insulation.

(a)

Temp (C)
20.00 20.75 21.50 21.25 23.00

(b)

Temp (C)
20.00 20.75 21.50 21.25 23.00

Figure 28. Calculated temperature contours and velocity vectors for Model B, with radiation modeling
(a) without insulation and (b) with insulation.

4.4 Lumped-Parameter Analysis

All of the CFX calculations for Models A and B above were 2-D, an assumption which enables simplification (and speedup) of the calculations compared to full 3-D modeling. The 2-D assumption was chosen because it is conservative in the sense that the actual thermal effects of the waste heat generated in the e-bay are expected to be less severe than those calculated, because the third dimension, which was not modeled, may provide additional pathways of airflow and walls for cooling. For Model B, this conservative assumption led to an OSS air temperature rise of 1–2°C, as shown by the CFX results discussed above. This relatively high temperature rise warrants additional analysis. One approach is to extend Model B to 3-D and perform more complex CFX calculations. This method was not pursued. Rather, a simpler lumped-parameter analysis is presented here, with the purpose of assessing how conservative the 2-D CFX results are for Model B by ignoring the OSS walls in the third dimension.

In this lumped-parameter study, it is assumed that the OSS air volume and the inner walls are at a single temperature (i.e., boundary layers and thermal gradients at the walls are ignored). This temperature is denoted as an increase from the ambient (20°C), namely ΔT_{OSS} . This temperature rise is determined by the balance of the energy coming in from the bottom wall, q_{in} , against the loss of energy through the vertical walls by natural convection of the outside air, q_{out} . The CFX calculations for Model B indicated that q_{in} is about 1.0 W. The rate of energy lost, q_{out} , can be related to ΔT_{OSS} as follows:

$$q_{out} = hA_{total}\Delta T_{OSS}, \quad (13)$$

where h is a heat transfer coefficient for natural convection at the temperature difference ΔT_{OSS} , and A_{total} is the total vertical wall area, which includes the side and end walls. The side walls are defined as those that are parallel to the forced flow in the e-bay, which were ignored in the 2-D CFX calculations. Therefore,

$$A_{total} = A_{sides} + A_{ends}, \quad (14)$$

$$A_{sides} = 7.083 \text{ m}^2, \quad (15)$$

$$A_{ends} = 1.118 \text{ m}^2, \quad (16)$$

$$A_{total} = 8.201 \text{ m}^2. \quad (17)$$

The heat transfer coefficient h depends on ΔT_{OSS} through a correlation involving the Rayleigh number, which is given in Eqs. 11 and 12. By requiring that q_{out} be equal to q_{in} , which is known, and explicitly writing h as a function of ΔT_{OSS} in Eq. 13, we have

$$h(\Delta T_{OSS}) A_{total} \Delta T_{OSS} = q_{in}. \quad (18)$$

The only unknown in the above equation is ΔT_{OSS} . However, an iterative procedure is required for its solution because the function $h(\Delta T_{OSS})$ as given by Eqs. 11 and 12 is nonlinear. However, starting from an initial guessed temperature difference of 1°C, one can easily obtain a converged solution for the set of Equations 11, 12, and 18 within several iterations. The solution is

$$\Delta T_{OSS} = 0.138^{\circ}\text{C}, \quad (19)$$

and the corresponding values for Ra_L and h are 6.5×10^6 and $0.886 \text{ W/m}^2\text{-}^{\circ}\text{C}$, respectively.

The temperature rise inside the OSS calculated here, at 0.138°C , is considerably lower than that calculated in the 2-D CFX Model B (which gives $1\text{--}2^{\circ}\text{C}$). However, the present lumped-parameter analysis under-predicts the OSS temperature because it ignores the inner wall boundary layers. In reality, the vertical walls are cooler than the bulk air in the OSS, thus lowering the natural convection heat loss capability (driving force) at the walls. Therefore, one cannot conclude based on this analysis that the OSS temperature rise is as low as 0.138°C . However, the lumped-parameter method is still useful in providing an estimate of the effect of ignoring the two side walls in the CFX calculations in Model B, as discussed in the following.

Comparing Eqs. 15 and 17, one can see that the side wall area represents 86% of all the available vertical wall area for natural convection heat transfer to the outside air. This large fraction of the surface area was ignored in the 2-D CFX calculations. To estimate the effect of leaving out the side wall area, the above lumped-parameter study is repeated using only the end wall area. In other words, Eq. 18 is solved with a slight change in variables to give

$$h(\Delta T'_{OSS})A_{ends} \Delta T'_{OSS} = q_{in}. \quad (20)$$

where $\Delta T'_{OSS}$ represents the temperature rise if only the end walls are included for heat transfer. Solution of Eq. 20, in conjunction with Eqs. 11 and 12, gives

$$\Delta T'_{oss} = 0.683^{\circ}\text{C}. \quad (21)$$

Now the effect of only having the end walls available for natural convection heat transfer can be quantified by the following factor

$$F_{2D} = \Delta T'_{oss} / \Delta T_{oss} = 5.0. \quad (22)$$

If one assumes that this conservative factor for the temperature rise also applies approximately to the CFX calculations, then one can estimate that the 2-D Model B results are also conservative by roughly a factor of five (as a result of ignoring the side walls). If all three dimensions are included in the modeling, the OSS air temperature rise will be five times less than 1–2°C, equal to 0.2–0.4°C.

The temperature rise can be further reduced by insulating the steel bottom wall of the OSS, across which escaped waste heat from the e-bay below enters. Without insulation, the temperature drop across the 0.318-cm-thick steel wall is of the order of 10^{-5} °C, which is negligible, for $q_{in} = 1.0$ W across the OSS bottom wall surface area of 2.428 m². If foam-board insulation with a thickness of 0.635 cm is applied to the bottom steel wall, then a temperature drop of 0.26°C will occur across the insulation for the same heat flux. Therefore, insulating the bottom wall of the OSS can reduce the temperature rise in the OSS air by up to 0.26°C.

4.5 Summary of PAM Design Recommendations

Based on the Model A results, the 20.28°C contour plume height is 0.94 m, which can be reduced to 0.39 m if the sheet metal is insulated with 0.635-cm-thick foam-board material. Based on the results of Model B and related lumped-parameter analyses, the average temperature rise of the air inside the OSS is 0.2 to 0.4°C, depending on whether the sheet metal is insulated. If the bottom of the OSS is insulated with the same foam-board material, then the OSS air temperature rise can be reduced by up to 0.26°C.

It is recommended that the sheet metal covering the e-bays and the bottom wall of the upper OSS be insulated to mitigate the thermal effects of the escaped waste heat on the air above the PAM units and on the air inside the OSS.

References

- [1] Bernardin, J. D., Parietti, L., and Martin, R. A., 1998, "Thermal Issues Associated with the HVAC and Lighting System Influences on the Performance of the National Ignition Facility Beam Transport Tubes," Technical Report LA-13410-MS, Los Alamos National Laboratory, Los Alamos, NM 87545.
- [2] CFX 4.2 Flow Solver User Guide, 1998, Computational Fluid Dynamic Services, Harwell Laboratory, Oxfordshire OX11 0RA, United Kingdom.
- [3] 1996, Marks' Standard Handbook for Mechanical Engineers, Tenth ed., Edited by Avallone, E. A., and Baumeister III, T., McGraw-Hill Co., Inc., New York, NY.
- [4] Van Doormaal, J. P., and Raithby, G. D., 1984, "Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows," *Num. Heat Trans.*, Vol. 7, pp. 147-163.
- [5] Incropera, F. P. and DeWitt, D. P., 1985, "Fundamentals of Heat and Mass Transfer," 2nd ed., John Wiley & Sons, Inc., New York, NY.
- [6] Churchill, S. W., and Chu, H. H. S., 1975, "Correlating Equations for Laminar and Turbulent Free Convection from a Vertical Plate," *Int. J. Heat Mass Transfer*, Vol. 18, pp. 1323-1329.

Appendix

Table A.1 provides a summary of the CFX modeling information pertaining to the electronics rack models. In particular, the net power to the rack, inlet airflow velocity, maximum rack surface temperature, and mass and energy balance errors are given for each model. The numerically predicted temperature contours and velocity vector plots corresponding to the models listed in Table A.1 are presented in Figures A.1 through A.12.

Filename	Power(W)	Inlet Flow (fpm)	Reference	Heat Flux/2	High Temperature (K)	Separation Distances (m) (top, sides)	Mass Error	Energy Error (%)
Var2	Linear temp. profile 293-299 K	0.00E+00	Figure A.1	9.92E+00	2.99E+02	2.5, .06	-1.29E-05	0.010
Symrack50	5.00E+01	0.00E+00	Figure A.2	2.50E+01	3.04E+02	3.49, .15	-2.60E-06	0.400
Symrack100	1.00E+02	0.00E+00	Figure A.3	5.00E+01	3.12E+02	Above model limit, .25	9.00E-06	0.670
Symrack250	2.50E+02	0.00E+00	Figure A.4	1.25E+02	3.30E+02	Above model limit, .35	6.40E-05	-0.108
Symrack500	5.00E+02	0.00E+00	Figure A.5	2.50E+02	3.57E+02	Above model limit, .38	6.70E-05	0.112
Symrack1000	1.00E+03	0.00E+00	Figure A.6	5.00E+02	4.08E+02	Above model limit, .35	-2.90E-05	-0.003
Symrack100a	1.00E+02	6.25E+00	Figure A.7	5.00E+01	3.12E+02	4.35, .25	1.00E-06	17.090 *
Symrack250a	2.50E+02	6.25E+00	Figure A.8	1.25E+02	3.30E+02	4.55, .27	-2.00E-06	17.170 *
Symrack1000a	1.00E+03	6.25E+00	Figure A.9	5.00E+02	4.19E+02	Above model limit, .25	-2.60E+05	19.480 *
12.5fpmracks	1.00E+02	1.25E+01	Figure A.10	5.00E+01	3.12E+02	Above model limit, .20	-1.10E-05	2.500
12.5fpmracksb	2.50E+02	1.25E+01	Figure A.11	1.25E+02	3.30E+02	Above model limit, .30	-3.50E-05	7.554
12.5fpmracksc	1.00E+03	1.25E+01	Figure A.12	5.00E+02	4.19E+02	Above model limit, .30	-1.20E-05	12.220

* Caution: Error displayed in Symrack100a, Symrack250a, and 1000a does not meet acceptance criteria.

Table A.1. General electronics rack information for each case.

Figure A.1. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack with a linear temperature profile ranging from 20 - 27 °C, and no forced external airflow.

Figure A.2. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 50 W, a maximum skin temperature of 31 °C, and no forced external airflow.

Figure A.3. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 100 W, a maximum skin temperature of 39 °C, and no forced external airflow.

Figure A.4. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 250 W, a maximum skin temperature of 57 °C, and no forced external airflow.

Figure A.5. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 500 W, a maximum skin temperature of 84 °C, and no forced external airflow.

Figure A.6. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack with a leakage power of 1000 W, a maximum skin temperature of 135 °C, and no forced external airflow.

Figure A.7. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage of 100 W, a maximum skin temperature of 39 °C, and 0.032 m/s forced external airflow.

Figure A.8. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 250 W, a maximum skin temperature of 57°C , and 0.032 m/s forced external airflow.

Figure A.9. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 1000 W, a skin temperature of 146 °C, and 0.032 m/s forced external airflow.

Figure A.10. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 100 W, a maximum skin temperature of 39 °C, and 0.064 m/s forced external airflow.

Figure A.11. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 250 W, a maximum skin temperature of 57 °C, and 0.064 m/s forced external airflow.

Figure A.12. Numerical model (a) temperature contour and (b) velocity vector plots of the electronics rack thermal plume for a rack leakage power of 1000 W, a maximum skin temperature of 146 °C, and 0.064 m/s forced external airflow.