PHOENICS Newsletter

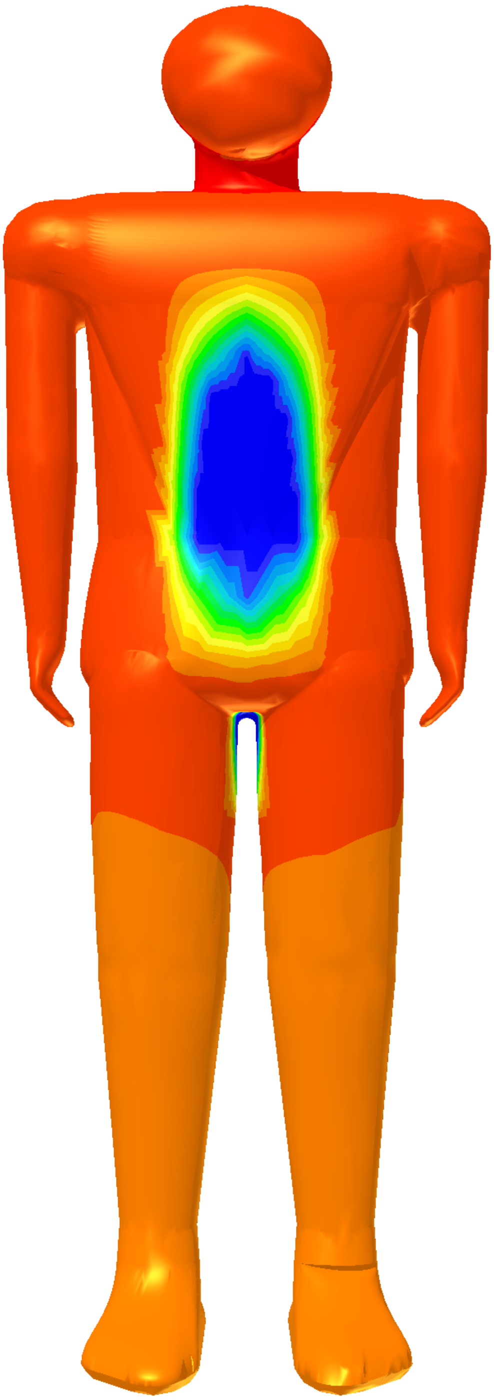


Table of Contents

Page

Concentration, Heat and Momentum Limited (CHAM)
Bakery House, 40 High Street, Wimbledon Village, London, SW19 5AU, England
Tel: +44 (0)20 8947 7651 Email: phoenics@cham.co.uk Web: www.cham.co.uk

Summer 2024

A Retrospective to Hannover Messe 2024 （Reprinted from the ZWSOFT Strategy and Branding Department）

From April 22nd to 26th, ZWSOFT showcased at the 2024 Hannover Messe. It is the largest industrial exhibition in the world and is considered a wind vane of global industrial technology development. Since 2008, ZWSOFT has been continuously exhibiting at the Hannover Messe, dedicated to promoting globally the innovative technological achievements of ZWSOFT's software suite.

CHAM, along with its parent company ZWSOFT, showcased at the Hannover Messe, promoting PHOENICS and introducing its applications in data centres and electronic cooling scenarios.

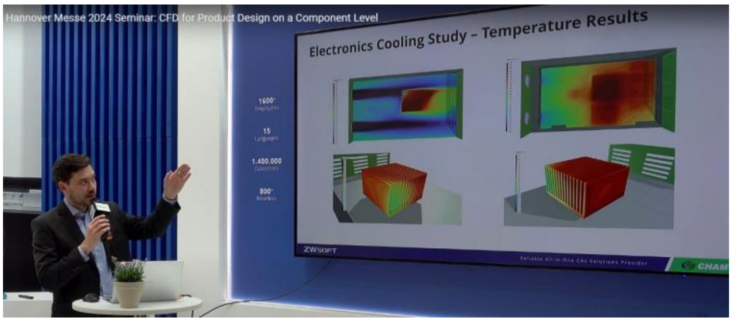


Fig1. Presentation on PHOENICS

Fig2. At the 2024 Hannover Messe, international business clients engage and experience at the ZWSOFT booth.

Since going global in 2004, ZWSOFT has spent 20 years continuously expanding and gradually building core technologies and solutions centered around independent 2D CAD, 3D CAD/CAM, and multi-disciplinary simulations in fluid, structural, and electromagnetic fields. Additionally, ZWSOFT has developed over 800 channel partners in countries and regions such as Germany, France, Poland, and Italy. It has also established localized subsidiaries in the United States, Japan, Vietnam, Malaysia, and the United Arab Emirates, forming a service capability that can quickly respond to user needs. This significantly enhances the application value of ZWSOFT's industrial software for global industrial enterprises. Currently, ZWSOFT serves more than 1.4 million industrial users in over 90 countries and regions worldwide.

Fig3. At the 2024 Hannover Messe, the ZWSOFT booth attracted numerous companies to stop by and visit.

Attracting Leading Talents to Build World-Class Software

Internationalization is a driving force for ZWSOFT's long-term and sustained development. On one hand, we have attracted numerous top-tier industrial software technology and R&D talents globally. On the other hand, through international exhibitions like the Hannover Messe, we have engaged in extensive exchanges with overseas customers, visiting and researching local enterprises and factories. This has enhanced our understanding of the international industrial software market and deepened our comprehension of overseas user needs at all stages, from R&D to sales. It better guides ZWSOFT's technological breakthroughs and product development.

With CHAM joining the ZWSOFT family, PHOENICS will appear at various international exhibitions, showcasing the new face of the world's first commercial general-purpose CFD software to a wide range of users.

Fig4. ZWSOFT staff at the 2024 Hannover Messe.

Natural Convection and Radiation Cooling of a Flow-Meter Heat Sink

Kathryn Potten[1], Timothy Brauner[1] and Hu Leilei[2]

CHAM, London,UK[1]; ZWSOFT, Guangzhou, China[2]

1. Introduction

Beijing Instrument Industry Group Co., Ltd. specializes in the design and manufacture of pipe flow meters, which are designed to measure the volumetric or mass flow rate of fluid within a pipeline. For high-temperature fluids, a metal heatsink with cooling fins is often installed between the flow meter and the pipeline. This is done to ensure that the fluid temperature entering the flow meter is not so high as to damage the unit. In heatsinks, heat transfer takes place via conduction, convection and radiation, and since in this case the fluid temperatures are high, and the device is cooled by natural convection, the amount of heat dissipated by radiation is likely to be a significant fraction of the total heat transfer rate.

There are several design parameters that can influence the performance of the heat sink, such as, for example, its length can be increased to promote more cooling with increase in temperature of the pipe-line fluid. Other design parameters include the material selection, the surface treatment, and the fin type, dimensions and pitch. PHOENICS has the potential to assist in the design process by making CFD simulations of the heatsink under different thermal conditions with differing design modifications. This article reports on some simulations made with PHOENICS which demonstrate its capability for predicting the thermal performance of this flow-meter heat-sink.

Figure 1 – Flow meter

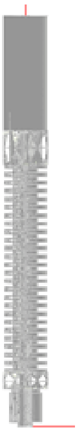


Figure 2 - Geometry used for CFD model in PHOENICS

2. The PHOENICS CFD Model

Modelling Assumptions and Strategy: The flow meter consists of a hollow, steel, water-filled tube with fins (the heatsink) which connects the flow meter, where measured flow is displayed, to the pipe housing the flow-measuring apparatus (see Figure 1). The section of pipe attached to the flow meter is then inserted into a larger pipe, through which the measured fluid moves with a temperature of 200^{o}C. The ambient temperature of the surrounding air is assumed to be 20^{o}C.

For the purposes of this demonstration simulation, a number of simplifying, modelling assumptions have been made. First, the geometry is simplified, keeping only the heatsink and omitting the flow meter (above the heatsink) and pipe (below the heatsink), as it is the temperature at the top of the heatsink that is of interest (Figure 2). This simplification renders the problem axisymmetric, and so two-dimensional calculations can be performed in a cylindrical-polar solution domain.

The Rayleigh number for this problem has been estimated to be about 2x10^{8}, which indicates that the flow is laminar, although perhaps undergoing transition to turbulence [2]. Nevertheless, the simulation is run as laminar, and a hand-calculated estimation of the radiative heat flux confirms that radiation needs to be taken into account, because it is significant compared to the convective heat flux. The present case is entirely buoyancy-driven free convection, where the flow is driven by a density difference.

The energy equation is solved with temperature as dependent variable, so as to predict the temperature entering the flow meter at the top of the heatsink. The thermal radiation is modelled by using the IMMERSOL radiation model. The emissivity of the steel is taken to be 0.3 based on experimental and numerical results at the expected temperatures [1].

Two additional simulations are performed without radiation, one with fins and one without them. The first allows an understanding of the effects of radiation and its importance in the model. The latter serves to provide a validation of free convection for a vertical cylinder at the specified Rayleigh number.

Physical Properties: Since the expected air temperatures range from 20 to 200^{o}C [2], the kinematic viscosity and thermal conductivity after calculated from Sutherland's law, and the air density from the ideal gas law.

The water inside the heatsink is modelled as a solid with the properties of water, as movement within the water is negligible and thus can be assumed stagnant. The properties of water are taken as constant because these do not change a significant amount over the expected temperature range.

Geometry: The heatsink is made up of two sections, with a total length of 25cm. The top section is a smooth steel cylinder, and the bottom a steel cylinder with 26 regularly-spaced annular fins (Figure 2). The water is modelled as a cylinder aligned centrally within the heatsink.

Boundary Conditions: The bottom of the flow meter is given a fixed temperature of 200°C to represent the fluid temperature inside the pipe. This is satisfied by creating a PLATE object at the bottom of the heatsink with the surface temperature set to 200^{o}C.

The bottom of the fluid domain uses a heat-insulated wall boundary (to represent the top of the pipe), and the remaining outer surfaces use open fixed-pressure boundary conditions, where air may leave or enter the solution domain to satisfy local continuity. The external ambient temperature is set to 20^{o}C.

3. Solution Domain and Mesh

The solution domain is taken to be a little over twice the height of the heatsink, with 4 times the diameter of the heatsink in the radial (Y) direction, corresponding to a cylindrical-polar grid of 0.1 radians in the X direction, radius (Y) 0.1m, and height (Z) 0.58m. The grid uses 61x484 cells, with sufficient cells in the Z direction to allow 5 cells per fin, and another 5 between each fin on the heatsink (Figure 3).

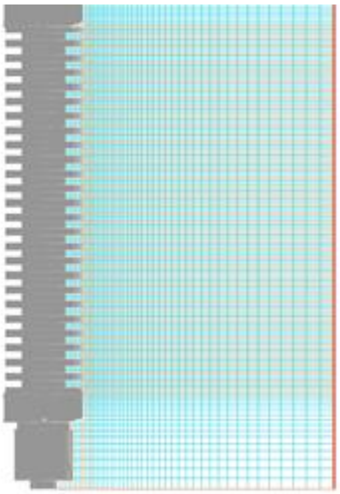


Figure 3 – Mesh distribution around the heatsink fins

This should be enough cells to resolve adequately the heat transfer around the fins, and capture some of the flow between them. No grid sensitivity study has been conducted for this demonstration exercise.

The solution is made in parallel mode on 4 cores using the CGRS solver for all variables with the AMG preconditioner for the P1, TEM1 and T3 equations, and a PBP preconditioner for the momentum equations.

4. Results and Discussion

The maximum temperature predicted inside the heat sink is 198°C, which is found at the bottom of the heatsink where it meets the pipe at which the 200°C boundary condition is specified. As shown in Figure 4, the temperature gradually decreases towards the top end of the heatsink, where the flow meter would be situated. At the top of the heatsink, where the heatsink meets the flow meter, the temperature is 53°C.

Table 1 presents results from the three runs: case 1, finned with radiation; case 2, finned without radiation; and case 3, smooth without radiation. The Nusselt number (Nu) was estimated for each case, and case 3 compared fairly well with correlations for a smooth, vertical cylinder in the literature. The total heat transfer coefficient (h), as well as heat fluxes (convective Q_c, radiative Q_r and total Q_t) and temperature at the top of the heatsink are shown. For the estimation of h and Nu, the total area over the fins was used together with the average surface temperature over the heatsink. Figure 5 shows the velocity contours predicted for Case 1.

Table 1 - Nusselt number, heat transfer coefficients, heat fluxes and temperature at top of heatsink

	Nu	h (W/m²K)	Q_c (W)	Q_r (W)	Q_t (W)	T at heatsink top (°C)
Case 1 – Fins + radiation	62.79	6.59	0.1206	0.1205	0.2411	53.19
Case 2 – Fins	31.22	3.28	0.1693	0	0.1693	78.22
Case 3 – No Fins	63.70	6.69	0.2577	0	0.2577	109.39
Data [2-5]	67.97-74.79	7.14-7.85	-	-	-	-

The temperature contours show that the heatsink is behaving as expected, with the heatsink cooling towards the top, as heat is convected away from the fins. A comparison of the runs with and without fins shows the importance of the fins in reducing sufficiently the temperature at the top of the heat sink, with a difference of around 30°C seen at the top of the heatsink. Comparing with and without radiation, it is clear that radiation plays a significant role in the cooling, with a difference of 25°C in temperature at the top of the heatsink, and the heat flux due to radiation being the same as that due to convection. Most of the heat loss due to radiation occurs near the base of the heatsink, where the temperature is highest. This has allowed a reduction in temperature from 200°C to 53°C, enough that the flow meter will not be damaged. This demonstrates that the heatsink is working properly and is consistent with true conditions.

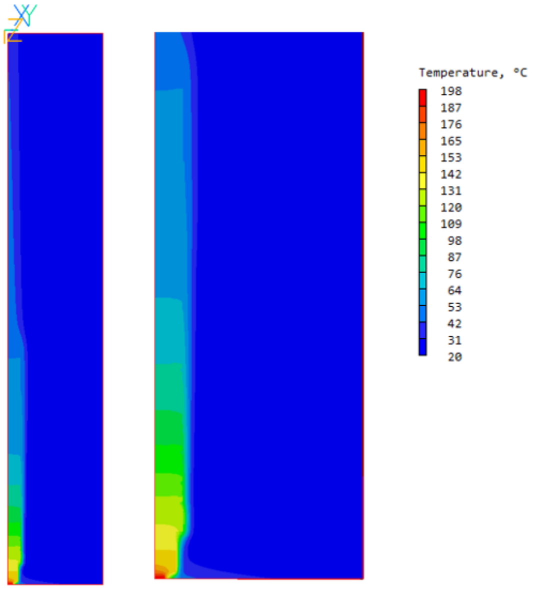


Figure 4 – Case 1 – Temperature contours

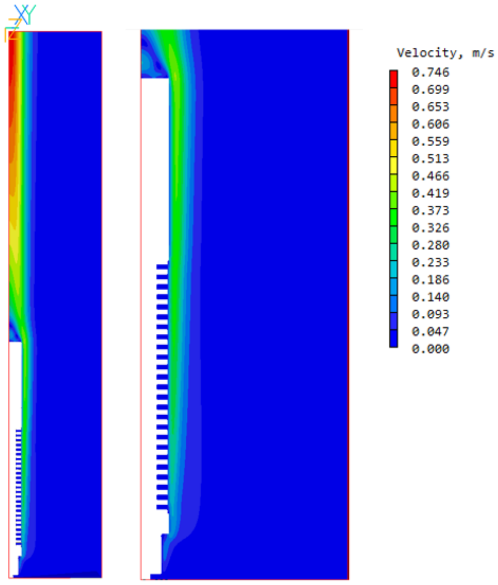


Figure 5 – Case 1 – Velocity contours

5. Conclusions

PHOENICS has been used to create a simplified CFD model of the heatsink of a pipe flow meter, in order to predict the temperature at the top of the heatsink, where it meets the flow meter. The heat-dissipation analysis has confirmed that for this simplified case, the heatsink can provide sufficient cooling to prevent damage to the flow meter. It has been shown that radiation is important in this case and cannot be neglected. The simplified CFD model could be used to investigate the effects of the fin geometry on the thermal performance of the heatsink. In addition, effects of length and diameter of the tube could be explored. Further simulations could be conducted using more complex geometry and boundary conditions, taking into account the curvature and surface temperature of the pipe itself, as well as the flow meter above the heatsink, which would all have effects on the flow and heat transfer.

6. References

1. Sadiq, H., Wong, M. B., Tashan, J., Al-Mahaidi, R. & Zhao, X. L. *Determination of steel emissivity for the temperature prediction of structural steel members in fire*. J. of Materials in Civil Engng.;25(2), 167-173, http://dx.doi.org/10.1061/(asce)mt.1943-5533.0000607, (2013).
2. Ozisik, M.N., *Heat Transfer, A Basic Approach*. New York: McGraw-Hill; (1985)
3. Popiel C.O., Wojtkowiak J., Bober K. *Laminar free convective heat transfer from an isothermal vertical slender cylinder.* Experimental Thermal & Fluid Science. 32:607-613. https://doi.org/10.1016/j.expthermflusci.2007.07.003, (2007).
4. S.W. Churchill, H.H.S. Chu. *Correlating equations for laminar and turbulent free convection from a vertical flat plate*. Int. J. Heat Mass Transfer; 18:1323–1329, (1975).
5. T. Cebeci. *Laminar-free-convective-heat transfer from the outer surface of a vertical slender circular cylinder*, in: Proc. 5th Int. Heat Transfer Conf., vol. 3. Paper NC 1.4, Tokyo; pp. 15–19, (1974).

Temperature analysis in a warehouse using a thermal insulation sheet

Toshiyuki Suzuki, CHAM Japan

1. Introduction

BX Tempal is a building-materials manufacturer, whose main product is a thermal-insulation sheet called "Haru Cool." CHAM Japan was requested by BX Tempal to perform a thermal analysis of a warehouse by using PHOENICS-FLAIR to predict the internal air-flow patterns and temperature distribution. The purpose of this thermal analysis was to express visually the beneficial effects of "Haru Cool", and to provide promotional materials for the manufacturer's sales activities.

"Haru Cool" is an indoor radiant-heat shield that uses aluminum foil, as indicated in Figure 1. The shield is lightweight, low cost, and easy to process. When the roof of a building is heated by sunlight (near infrared radiation), "Haru Cool" can be installed beneath the roof, where it acts as a thermal barrier by reflecting the radiant heat from the roof. This barrier suppresses the rise in indoor temperature, which in turn protects the goods and materials stored in the warehouse below.

(a) Thermal insulation sheet "Haru Cool."

(b) Roof shape and solar panels

(c) Luggage inside the warehouse

Figure1. The warehouse considered

2. PHOENICS-FLAIR simulations

a) Simulation Details

Numerical simulations are made with and without a thermal insulation sheet for the upper-corner room of an existing two-story warehouse. Figure 2 shows the solution domain for the warehouse to be analysed. The warehouse has the following dimensions: 45.6m x 27.6m x 6.325m, and it is made of corrugated galvalume-steel sheets, with solar panels installed on top of the roof. In Figure 2, the pink areas of the roof are the parts exposed to sunlight. The north and west sides are the exterior walls and these together with the roof panels provide heat transfer at 35°Cplus the Tokyo average solar-radiation flux of 176 W/m^2. There are eight cardboard boxes in the room, each measuring 8m x 8m x 5.5m.

The emissivities of the roof, outer wall, inner wall, floor, and luggage are 0.7, 0.8, 0.9, 0.54, and 0.9, respectively. "Haru Cool" is placed directly under the roof and in PHOENICS-FLAIR it is modelled as an aluminium THIN-PLATE object with a specified thickness of 4mm and an emissivity of 0.09. The IMMERSOL radiation model, the Chen-Kim k-ε two-equation turbulence model, and the Boussinesq approximation for buoyancy are utilised in the CFD simulations.

In order to express thermal comfort intuitively through the simulation results, a human body, with dimensions 0.5m x 0.3m x 1.7m, and a presumed surface emissivity of 0.9, was placed in the centre of the warehouse. In the simulations, the following values were used for deriving thermal comfort indices [1] from the CFD results: internal heat source 80W, metabolic rate 1.2 met (1 met = 58.2 W/m^2), and amount of clothing 0.6 clo (1 clo=0.155 m^2 ^{O}C/W).

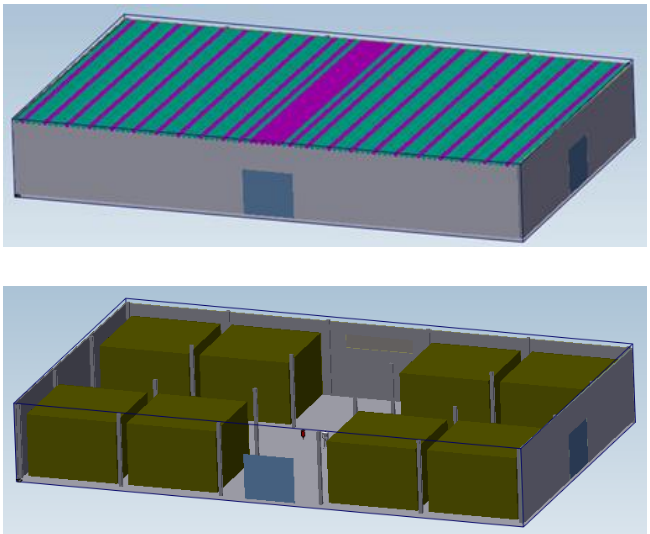
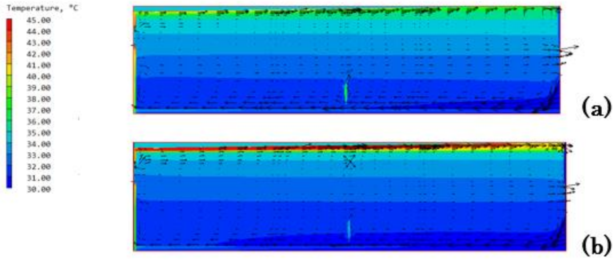


Figure 2: Simulation Model and Solution Domain

b) Results and Discussion

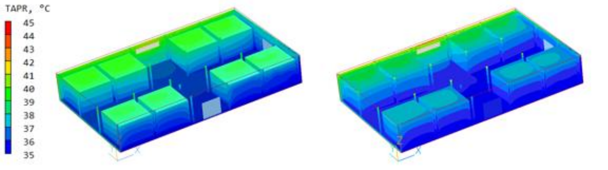
Figure 3 shows the temperature distribution across the central cross-section of the warehouse, without “Haru Cool” (a), and with “Haru Cool” (b). With "Haru Cool", the temperature in the gap between "Haru Cool" and the roof is high, and the indoor temperature is slightly lower.

Figure 4 shows the surface temperature distribution of the solid walls without “Haru Cool” (a) and with “Haru Cool” (b). If there is no "Haru Cool", it means that the cargo in the warehouse is being overheated due to radiant heat from the roof.



(a) without “Haru Cool” **(b) with “Haru Cool**

Figure 3 Temperature distribution at the central cross-section



(a) without “Haru Cool” (b) with “Haru Cool

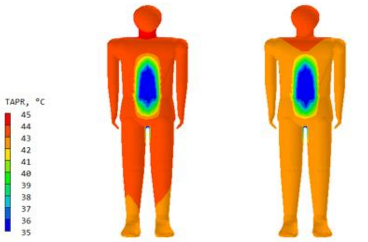
Figure 4 Surface temperature distribution of solid walls

Figure 5 shows the surface distribution of the apparent temperature, T_A, which is the temperature perceived by a human as a result of the combined effects of air temperature, humidity and air speed. The apparent temperature is computed from:

$$T_A = T_a + 0.348e - 0.7w_s + \frac{0.7Q}{w_s+10} - 4.25 \quad (1)$$

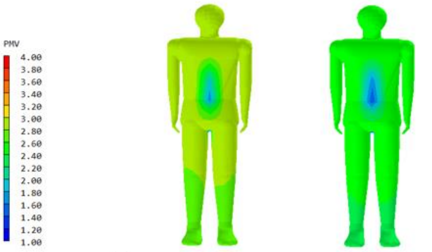
where T_a is the dry-bulb temperature; e the water-vapour partial pressure; w_s the air velocity; and Q the radiant heat flux.

Figure 6 shows the surface distribution of predicted mean vote (PMV) on the human occupant. The PVM, which is calculated as defined in ISO7730 [1], is a thermal-comfort index that aims to predict the mean value of votes of a group of room occupants on a seven-point thermal sensation scale, where -3 is cold, 0 is neutral and +3 is hot. The PVM considers the heat balance of the human body and it is calculated from a number of parameters, including the air temperature, the radiant temperature, the humidity, the air velocity, the metabolic rate and clothing insulation.



(a) without "Haru Cool" (b) with "Haru Cool"

Figure 5 Apparent Temperature （T_A）



(a) without "Haru Cool" (b) with "Haru Cool"

Figure 6 Predicted mean vote (PMV)

Conclusion

Thermal simulations were carried out with PHOENICS-FLAIR for an existing warehouse, and the effects of installing heat-shielding sheets were investigated with boundary conditions of solar radiation, temperature, and humidity in the actual environment. The CFD study confirmed that by installing a heat-shielding sheet to reflect thermal radiation, the room became more comfortable. Based on this visualization, construction work for "Haru Cool" was scheduled for this warehouse.

4. References

ISO 7730: 2005 Ergonomics of the thermal environment – Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria.

How to use the "HOTBOX Parts" in Electronic Cooling Simulation?

Rongkang Xu and Isaac Wang

On September 2, 2024, the PHOENICS series product HOTBOX2 will make its debut. This software is designed for the electronics industry, aiming to help engineers optimize the thermal performance of their electronic products. To simplify and accelerate the electronic cooling design process, HOTBOX2 provides a quick design module that allows the creation of predefined models called "HOTBOX Parts". Users can select and use these "HOTBOX Parts" from the library without starting from scratch. The "HOTBOX Parts" are parameterized, enabling users to modify the size, material properties, and other characteristics of the parts through simple parameter adjustments, thus achieving fast and accurate designs.

Types of "HOTBOX Parts":

The "HOTBOX Parts" library offers a wide range of predefined component types, covering various common electronic components and structures, including heat sources, fans, heat sinks, PCBs, chips, enclosures, and more. Heat sources are used to simulate the heat generated by electronic components. Fans include different types such as axial and centrifugal, simulating the effects of cooling fans. Heat sinks can be selected from finned or plate-type heat sinks to enhance heat-dissipation efficiency. The different types of components can be used to ease the setup of cases with complex thermal management requirements.

Creation of "HOTBOX Parts":

In the main view area of HOTBOX2, on the left side (as shown in Fig.1), users can select suitable components from the "HOTBOX Parts" library and add them to the model with a single mouse click.

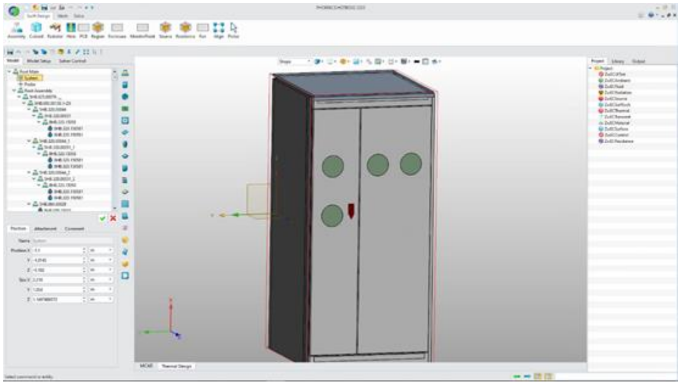
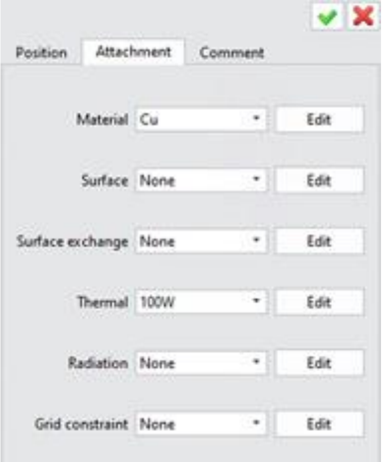


Figure.1 HOTBOX Part Panel

Settings of “HOTBOX Parts”:

During setup, in order to match the design specifications, the user must define various parameters and properties of the “HOTBOX Part”, such as dimensions, material properties, thermal properties or radiation related properties, etc. (as shown in Fig.2). For this purpose HOTBOX2 provides a standard model library with parameters supplied by manufacturers. This data can be used directly to define the properties of a HOTBOX Part, making setting up a model as close to reality as possible quick and easy. Depending on the background of the application or the stage of the design process, different requirements for the model may be prioritised by the user; these may affect the level of detail in the model setup, the desired accuracy of the simulation outputs or the speed at which results can be produced. HOTBOX Parts can be used to streamline the process of setting up a model and also adapting it to the user’s current needs.



Material: Cu
Surface: None
Surface exchange: None
Thermal: 100W
Radiation: None
Grid constraint: None

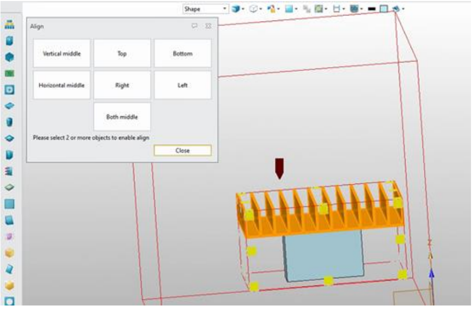
Figure.2 Settings of “HOTBOX Parts”

Name: 100W
Thermal Model: Conduction
Total Power: 100 W
Transient Attribute: None

Figure.3 Project tree

Establishing a Thermal Management Model:

By dragging and snapping HOTBOX Parts, or using the alignment functions (top/bottom, left/right, horizontal centre, vertical centre), users can place them in the appropriate positions and connect them according to the actual design, such as mounting a heat sink above a heat source (as shown in Fig.4).



Please select 2 or more objects to enable align

Figure.4 Alignment panel

By using additional HOTBOX Part commands like array, mirror, rotate, etc., designers can quickly construct complex thermal management models composed of many components. These commands help streamline the process of model refinement, and allow users to rapidly perform optimization analysis on the model’s thermal performance.

HOTBOX Parts offer significant advantages in electronic thermal simulation, greatly simplifying the modeling process, improving simulation efficiency and accuracy, and providing greater design flexibility and optimization space. By effectively utilizing HOTBOX Parts, engineers can more efficiently conduct thermal management design, enhancing the thermal performance and reliability of electronic products. These advantages make HOTBOX Parts an indispensable tool in modern electronic thermal simulation.

Forced-convection air cooling of an electronics cabinet

Kathryn Potten and Timothy Brauner

1. Introduction

A case study has been conducted in PHOENICS of the cooling of an electronics cabinet through forced convection. The cabinet is a typical server unit, inside of which is a PCB board, on which rest a number of heat-emitting chips such as would usually be found in a computer, plus a finned heatsink. A constant stream of air is forced into the cabinet through two inlets to cool the components, before exiting through an array of outlet slots on the opposite side of the cabinet.

A simple CFD simulation was run in PHOENICS to study the air flow and heat distribution inside the cabinet, with especial interest on the temperatures of the components. A number of simplifying assumptions were made for this case study.

2. The PHOENICS CFD model

Modelling Assumptions and Strategy

The energy equation was solved for temperature without radiation, as the effects of radiation were deemed to be small enough to neglect for an initial study. The flow is turbulent, and the Chen-Kim two-equation k-ε turbulence model is used in the simulations. However, an estimation of the Reynolds number between the fins of the heatsink suggests laminar flow, and so the Chen-Kim model is not strictly valid. For this simulation, this complication was ignored for simplicity, and further work would be needed to model the flow more accurately between the fins.

The flow is assumed steady, and the cabinet casing is assumed adiabatic, which means all the heat must leave the cabinet through the outlet slots. Buoyancy effects, which may be present locally in some regions, are ignored in the present simulations.

Physical Properties

The air inside the cabinet and at the inlets is at an ambient temperature of 20^{o}C. The case of the cabinet and heatsink are aluminium, and the chips and PCB board properties were specified as new materials through the PHOENICS PROPS file. Their properties can be seen in Table 2.

Name	Aluminium (Case/heat sink)	PCB Board	Chips
Density (kg/m³)	2719	1000	1000
Specific Heat Capacity (J/kgK)	871	150	100
Thermal Conductivity (W/mK)	202.4	30	15

Table 2 - Component Properties

Geometry and Solution Domain

The electronics cabinet case has 2 round inlets on the front and a series of slotted outlets on the rear. Inside the case, various heat-emitting chips, ranging from 1W to 20W, are spread across the PCB board. The heatsink consists of a base and a number of vertical fins with small channels between them, and these are placed directly on top of the 20W chip. The set-up in PHOENICS is shown in Figure 1. Aside from the ends of the cabinet, which were imported from CAD objects, the geometry was built from the basic geometrical shapes available in PHOENICS. For simplicity, a single INLET object was placed at the end of the domain outside the case to simulate the forced flow into the system. An individual OUTLET object was placed behind each slot from the case to allow the average temperature at each slot to be recorded.

The size of the cabinet is approximately 20 cm x 11 cm x 4 cm.

Boundary Conditions

The inlets were defined to have a fixed, uniform velocity of 2m/s, with a turbulence intensity of 5%. The air entering the domain at the inlets is set to an ambient temperature of 20^{o}C.

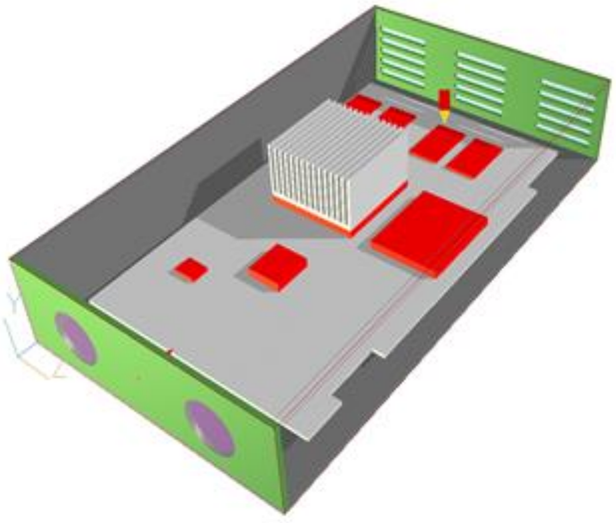


Figure 1 - Geometry set-up in PHOENICS

Mesh

A Cartesian immersed-boundary (SPARSOL) mesh of 140x40x140 cells was used. This mesh is somewhat coarse for the problem; and further grid refinement was not performed for this study. In order to have a sufficient number of cells in the thin channels between the fins, at least 2 cells per fin (width-wise), and three cells inside the channels are required. Otherwise, the solution between the fins is entirely dependent on the wall functions and will not be realistic.

3. Results and Discussion

The average predicted temperature within the cabinet is 49.8^{o}C, with a peak temperature of 99.5^{o} achieved within the domain. This high point is at the chip with 20W heating power, which is the hottest component showing an average temperature of 96.3^{o}C. The average exit temperature at the outlets is 41.5^{o}C, from an initial ambient temperature of 20^{o}C.

Two streams of fast air emanate directly from the two inlets where cool air is forced into the cabinet. Areas outside of the direct line of airflow from these inlets, have low velocities, and thus show significant heating compared to the continually refreshed inlet streams.

Due to the nature of the outlet configuration, with multiple thin slots, some of the air is unable to leave the cabinet and recirculates back into the main flow (Figure 3).

The heatsink shows an average temperature of 95.2^{o}C, slightly below that of the 20W chip that it rests on (Figure 4). On the side with stronger airflow (that in direct line from the leftmost inlet) we see lower temperatures of 40-75^{o}C, with the opposite side (which is out of the direct line of flow from an inlet, with velocity close to zero) showing the higher temperature of nearly 100^{o}C. This is as would be expected because the cooler side has a constant flow of cooler air passing alongside the fins.

The PCB board has, as would be anticipated, its maximum temperature 99.4^{o}C where the hottest chip sits. However, the average temperature across the board is significantly lower, at 77.8^{o}C. This is due to the front end of the board being cooled by the constant flow of air from the inlets.

Pressure is largely constant within the flow field, with a region of stagnation pressure appearing where the flow impacts the heatsink, and a second near the outlets.

4. Conclusion

The PHOENICS solution for this study of forced-convection air cooling of an electronics cabinet shows an overall flow pattern largely as expected. Two main streams of air flow through the cabinet, and are interrupted by contact with the heatsink, showing some recirculation near the outlets (due to the small area of the outlet slots) and in the wake of the heatsink. The hottest parts of the domain are at the chip with the highest heating power, and at the aluminium heatsink placed above it. This shows that the heatsink is working as designed, drawing heat away from the chip on which it is placed.

Further work could include a radiation model such as IMMERSOL to investigate the effects of radiation on the cooling. Alternative turbulence models could be investigated to improve the solution between the fins, such as the LVEL model which may prove better at capturing flow in small channels. Finally, a finer mesh would allow better resolution between the fins, as well as improving the accuracy of the heat transfer estimation and the pressure drop across the heatsink.

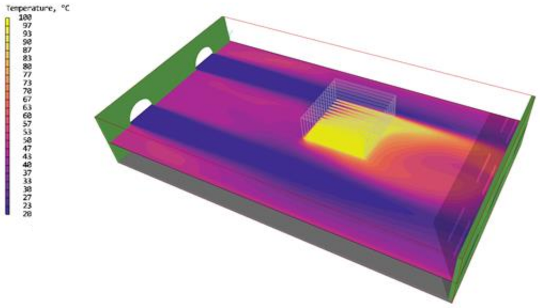


Figure 2 - Temperature contours

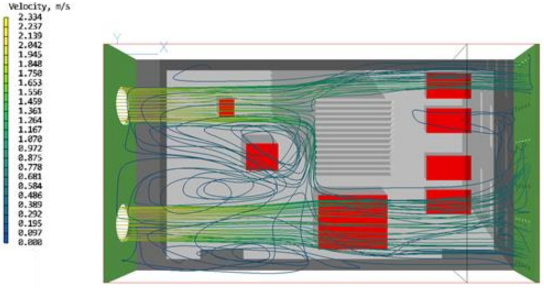


Figure 3 - Velocity streamlines.

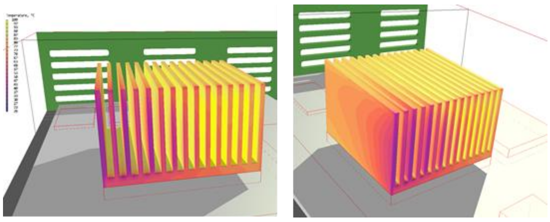


Figure 4 - Heatsink surface temperature contours

News from CHAM:

We at CHAM would like to welcome our two latest recruits into the CHAM family:

Kathryn Potten
(Junior CFD Engineer)

Nimi Uranta
(Business Development Manager)

News from CHAM Japan and ZWSOFT:

June 19,20,21 CHAM Japan exhibited PHOENICS at the 36th Design Manufacturing Solution Expo(DMS). DMS is one of Japan's leading exhibitions for the design and manufacturing industries, bringing together CAD and simulation software as well as a wide range of equipment and devices. This time, we exhibited together with ZWSOFT's 2D and 3D CAD

Contact Us:

CHAM's highly skilled, and helpful, technical team can assist in solving your CFD problems via proven, cost-effective, and reliable, CFD software solutions, training, technical support and consulting services. If YOU have a CFD problem why not get in touch to see how WE can help with the solution?

Please call on +44 (20) 89477651, email sales@cham.co.uk or check our website www.cham.co.uk. For PHOENICS on the Cloud (PHOENICS-OTC) call us or contact phoenics.cloud@cham.co.uk

See us on social media sites shown below:

Concentration Heat and Momentum Limited (CHAM)
Bakery House, 40 High Street
Wimbledon Village
London SW19 5AU, England