Proceedings of the 4th WSEAS Int. Conf. on HEAT TRANSFER, THERMAL ENGINEERING and ENVIRONMENT, Elounda, Greece, August 21-23, 2006 (pp122-126)

Aerothermal design of electronical device cabinet

RESUL ACIKYOL, FIRAT O. EDIS, M. FEVZI UNAL
Faculty of Aeronautical and Astronautical
Istanbul Technical University
TURKEY

Abstract: In this study, heat and fluid flow in a small cabinet is simulated using Computational Fluid Dynamics software FLUENT. Various cabinet designs are proposed by positioning the different type of exhaust fans, air separator sheets and cabinet air inlet in various sizes. Aerothermal analysis is performed for the proposed cabinets to reach desired flow conditions at the inlet of the devices placed in the cabinet. Promising cabinet design results obtained at the end of these analyses are given with a sample results for comparison in this article.

Key-Words: Aerothermal analysis, Heat and fluid flow simulation, Electronical device cabinet design

1 Introduction

In this study, a cabinet, shown in Fig. 1, containing a number of electronic devices is analyzed for heat and fluid flow. Heat is generated in devices and convected into the cabinet using cooling fans. The aerothermal analysis is performed using Computational Fluid Dynamics, being a reliable and efficient tool for such complex flow fields. Similar work is done by Cheong [1] and Gebremedhin [2], to predict airflow in indoor environments and to understand air conditioning effects. In this study, however, the computational domain is a small device cabinet. The design of the cabinet must ensure that fresh cooling air is supplied to all device inlets. Furthermore, the device exhaust flows at increased temperatures due to the heat generation in the devices, must be guided to the cabinet exits with minimal mixing to the cooling air of the devices. A measure for the performance of the cabinet is the temperature difference between the cabinet inlet and device inlets. A good design must provide minimal, preferable no temperature increase.

In this study, permitted increment in the average temperature values at the device's inlets are 3^{o}C for device-1 and no increment for both the devices 2 and 3. To reach these temperature increments at the device inlets various cabinet designs are proposed and heat and fluid flow in these cabinets are simulated. Obtained promising cabinet design and its details are given in the result section.

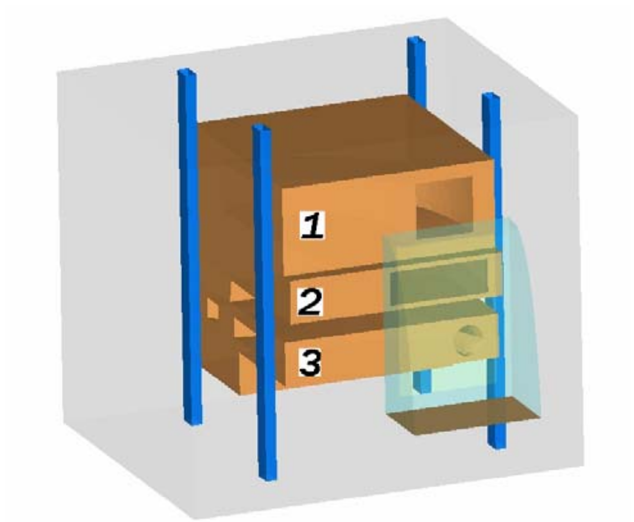

Fig.1 Cabinet and devices

2 Governing Equations

To simulate flow and heat transfer in a cabinet having numbers of devices, commercial software code Fluent 6.2.16 is used. The software utilizes finite volume approach, which is mostly used in CFD, addresses the problem by subdividing the computational domain in to a many subdomains where flow parameters are described. By using this finite volume code, Navier-Stokes equations are solved in the steady (time independent), incompressible (constant density) form, implicitly. Continuity, momentum and thermal energy equations in Cartesian coordinates for incompressible flow are in the following form [3]:

$$\frac{D\mathbf{u}}{Dt} = 0 \tag{1}$$

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \rho \mathbf{g} + \mu \nabla^2 \mathbf{u} \tag{2}$$

$$\rho \frac{De}{Dt} = -\nabla \cdot \mathbf{q} - p(\nabla \cdot \mathbf{u}) + \phi \tag{3}$$

In the equations written above, $\mathbf{u}$, p, ρ, e, $\mathbf{q}$, ϕ denote the velocity vector, pressure, density, internal energy, heat flux and source term, respectively.
Radiation and buoyancy effects are neglected in the simulations. On the other hand, effect of exhaust and cooling fans, heat sources and electromagnetic\dust filter on flow are taken in to account, details of modeling are given in the following subsections.

2.1 Fan Simulation

In the presented study, cooling fans and exhaust fans are modeled as 2-D surfaces. Mass flow rates produced by these fans are function of pressure difference (Δp) between two sides of it. If the pressure jump value vanishes, fans work in maximum performance. The relationship between the pressure jump and fan velocity is given with following equation.

$$\Delta p = \sum_{n=1}^{N} f_n v^{n-1} \tag{4}$$

where f_n are pressure jump coefficients and v is the magnitude of the local fluid velocity normal to the fan. To model the cooling and exhaust fans, the second and third order approximations in velocity (n=2, 3) are used in the presented study.

Three type of exhaust fans are used in the cabinet and its mass flow rate-pressure jump variation graphics are given in the Fig. 2. Velocity-pressure jump variation of cooling fans which is placed in the internal devices is given in the Fig.3. Device-1 and device-3 has one internal fan while the device-2 has three. Except the fan 1 and 2 of device-2 all the cooling fans behave differently (see Fig.3)

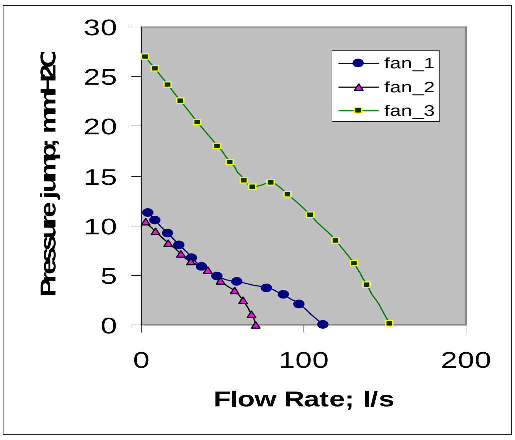

Fig. 2. Mass flow rate- pressure jump graphics for the exhaust fans

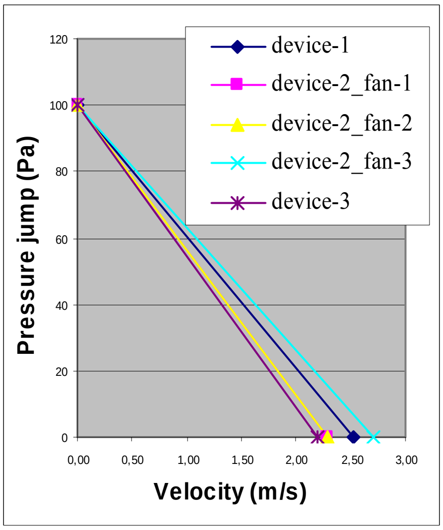

Fig. 3 Velocity-pressure jump graphics for the device cooling fans

2.2 Heat Source Simulation

The heat sources placed in the devices cause temperature increments in the cabinet. In this study, it is assumed that produced heat values are constants in time and the values used in the analysis are given in the Table.2.

Device	Produced heats values (kW/m^3)
Device-1_source	6521,739
Device-2_source-1	2083,333
Device-2_source-2	2083,333
Device-2_source-3	10237,389
Device-3_source	2688,172

Table 1 Produced heat values in devices

2.3 Porous inlet simulation

Electromagnetic/dust filter placed at the cabinet inlet is modeled as 2-D surface and fluid flow through this surface is modeled using the porous-jump approach available in the FLUENT. For the pressure jump value (ΔP) of a porous media, following equation is used in the computations:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \qquad (5)$$

where α is the surface permeability, C_2 is the pressure-jump coefficient and Δm is the thickness of the medium[4].

3 Computational Details

3.1. Geometrical details

The cabinet shown in the Figure 1 is a rectangular prism and its dimensions in millimeter are given in Table 2. Geometrical positions and sizes of three devices placed in the cabinet and sizes of the cooling fans and air inlet placed on the surfaces of these devices are also listed in this table. For all the cabinet flow simulated in this study, sizes and positions of the devices, cooling fans and device's air inlets remain same while the position, size and numbers of exhaust fans and electromagnetic/dust filters (air inlet) placed on the cabinet surfaces are changed to obtain desired flow conditions.

	Length	Depth	Height	Position
Cabinet	800	670	705	0,0,0
Device 1	435	450	176	120,544,197
Dev.1 fan	-	146	99	555,377,462
Dev.1 inlet	117	-	112	165,535,453
Device 2	435	483	87	120,568,329
Dev.2 fan-1	60	-	60	155,85,382
Dev.2 fan-2	60	-	60	215,85,382
Dev.2 fan-3	150	-	55	420,85,352
Dev.2 inlet	200	-	67	150,568,339
Device 3	435	459	89	120,535,417
Dev.3 fan	62	-	62	182,544,240
Dev.3 outlet	-	57	41	558,204,220
	-	160	41	558,444,220

Table 2 Dimensions and positions of cabinet and devices

Electromagnetic/dust filter is placed on the side wall of the cabinet while the exhaust fans are placed on the upper wall of the cabinet as shown in the Fig. 4 and Fig 5. Unstructured computational mesh shown in the Fig. 4 is one of the grids used in the computations. For all the cases simulated in this study, new computational meshes are generated using the software GRIDGEN [5]. A typical mesh has about 240000 tetrahedral elements.

It is assumed that heat sources are in the middle of the channels connecting the air inlet and outlet of devices can be seen in Fig. 5.

Reynolds number based on the cabinet height and average inlet velocity is in the range of 25000 to 45000 for the simulated cases. Time independent (steady) $k-\varepsilon$ turbulence model with standard wall functions is used in the computations. This turbulence model has been used by Cheong and Gebremedhin also for indoor aerothermal design problems. SIMPLE pressure-velocity coupling algorithm is used in the computations.

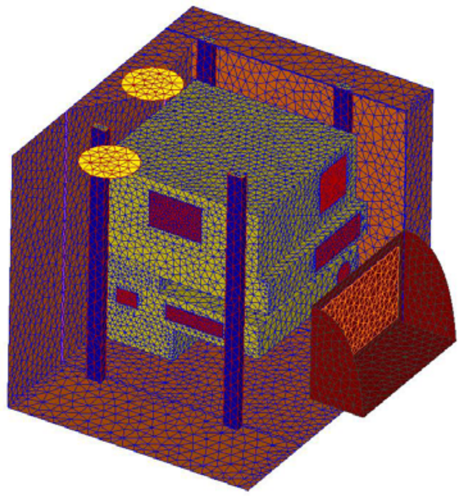

Fig. 4 Computational domain, grid

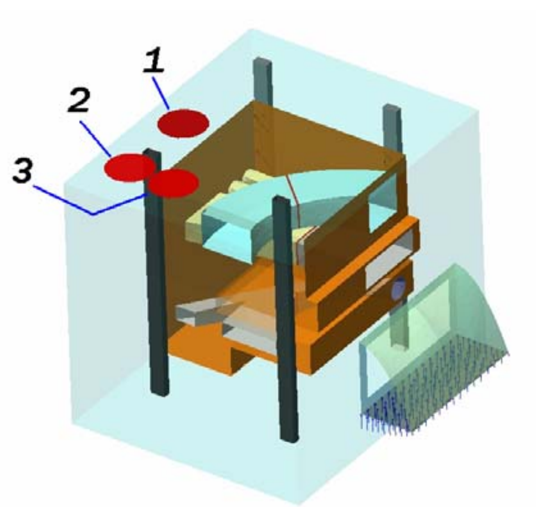

Fig. 5 Location of exhaust fans

3.2 Design Alternatives

To reach desired flow condition, position and size of the air inlet placed on the side wall of the cabinet can be changed. In the presented study, various flow cases are constructed by this way and these cases are analyzed. Sizes and positions of the air inlet for studied 6 cases are given in Fig. 6. Another cabinet design alternative that is taken into account is changing the number and position of the exhaust fan placed on the cabinet upper wall. In Fig. 5, one of the cases studied for this aim can be seen. In this case, three exhaust fans (in red) are located on the left hand side corner of the upper wall.

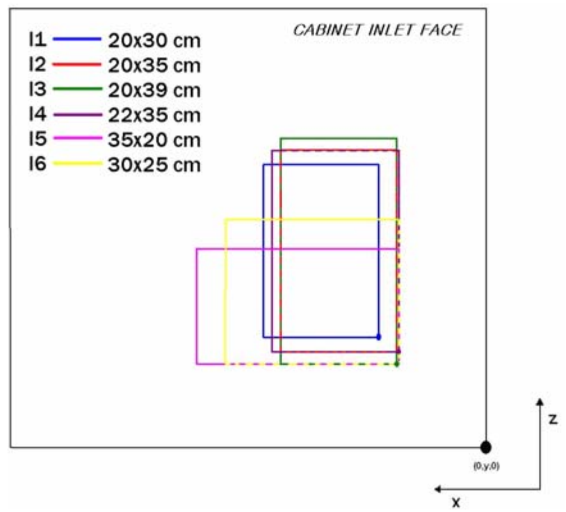

Fig. 6 Cabinet inlet size and location

Using metal sheet in the cabinet as an air separator to prevent mixing of fresh cool air with the heated flow in the devices is another cabinet designing alternative. Different flow cases are constructed for this aim by placing one or more separating sheets in the locations shown in Fig. 7.

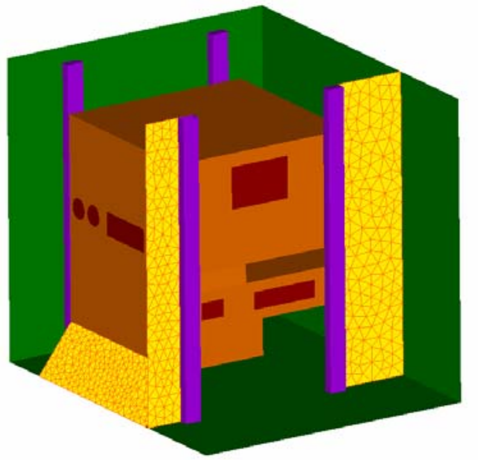

Fig. 7 Air separator sheets

4 Results

Systematically using combinations of three different ways in cabinet designing mentioned above, 60 flow cases are constructed and analyzed. Among all the studied cases, desired results are obtained for the case having two exhaust fans and air inlet placed on the cabinet side wall horizontally. Beside the aerothermal performance, financial requirements make the designs that have air separator sheets less preferred compared to the alternatives. On the other hand using wider air inlet for the cabinet causes higher financial requirements due to the increasing electromagnetic/dust filter dimensions. Another unwanted effect is the vibration produced by increased numbers of exhaust fans. Because of these restrictions, cases that have fewer exhaust fans and no air separator sheets are preferred to the other ones if desired flow conditions are satisfied for both cases. In the following sections one of the average and promising results is given in details.

4.1 Sample Result

A flow case is constructed by placing two fan_3 on the top wall and air inlet on the side wall. Air inlet dimensions are 20 cm in horizontal and 30 cm in vertical directions. Temperature increment relative to the cabinet inlet values are listed in the Table 3 for the flow entry of devices.

As seen in the table, temperature increment values for device-2 and 3 are higher than the desired values of 0. The increment in the temperature values at the device inlet is a result of wrong placed exhaust fans and smaller cabinet air inlet. In Fig. 8, path lines in the cabinet are plotted. As seen in the figure, cabinet air inlet can not support enough fresh air for device-3 which gets most of the air from heated exhaust flow of device-2.

Device No	Temperature increment
1	1
2	1
3	6

Table 3 Relative temperature increase at inlet of the devices

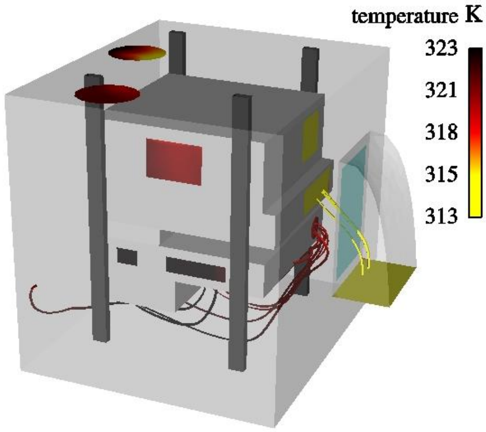

Fig. 8 Pathlines at device-3 inlet

4.2 Promising design alternative

Desired results are obtained simulating the flow in the cabinet having two exhaust fan_1 at the top wall and cabinet air inlet with dimensions of 35x25 cm in horizontal and vertical directions. Temperature increment values at the devices inlets are listed in Table 4. Obtained temperature increment values are acceptable for all the devices. The temperature increment in the device-1 inlet is a result of mixing of heated exhaust flow of this device with the fresh cool air supplied from the cabinet inlet. In Fig. 9, path lines are given for device-1 and mixing of cooling air with heated air can be seen.

Device No	Temperature increment
1	1
2	0
3	0

Table 4 Relative temperature increase at inlet of the devices

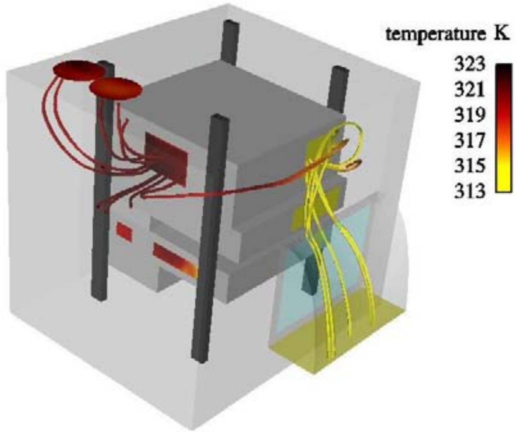

Fig. 9 Pathlines at device-1 inlet

5 Conclusion

In this study, heat and fluid flow in the small cabinet that has heat generating devices inside is simulated using commercial software code FLUENT. To determine the optimal cabinet design that has the permitted range of temperature increment values at the device's air inlets; number of flow cases are constructed by repositioning the cabinet air inlet/exhaust fans, changing the numbers of exhaust fans and the size of the cabinet air inlet. Aerothermal analyses are performed for all the cases and obtained promising design result is given with a sample one. Path lines in the cabinet are plotted for these two cases and effect of cabinet design on heat and fluid flow is discussed. With this study, it is shown that CFD is a reliable and efficient tool for such complex flow fields.

References:

[1] Cheong K.W.D., Development of a simplified technique of modeling four-way ceiling air diffuser, Building and Environment, 2001

[2] Gebremedhin K.G., Numerical simulation of flow field around a cow using 3-D body-fitted coordinate system, Journal of Thermal Biology, 2001

[3] P. K. Kundu, I. M. Cohen, Fluid Mechanics, Academic Press, 2002

[4] Fluent User Guide, Version 6.2, Fluent Inc.

[5] Gridgen User Guide, Version 15.08, Pointwise Inc