



CHAM Limited

Pioneering CFD Software for Education & Industry

PHOENICS Demonstration

Heat transfer in PCB cabinet

ABSTRACT

This report describes the application of the **PHOENICS** computer code to the simulation of the air flow and heat transfer processes occurring within a PCB cabinet. The CFD predictions were carried out at steady state and the influence of air velocity, and temperature is clearly shown.

1.0 INTRODUCTION

1.1 Purpose of the Report.

This report describes a demonstration case using **PHOENICS** to analyze the gas flow and heat transfer processes occurring within a PCB cabinet. The study was performed by **CHAM** on behalf of **TERMA** Ltd in order to provide insight to the capability of the CFD code in predicting such systems.

The use of CFD in conjunction with the existing design methods can provide:

- Understanding of the local air flow and heat transfer (which could not otherwise be obtained).
- Basis by which design modifications can be devised.
- The ability to assess design modifications without the need for slow and expensive building and testing design cycle.

The report provides brief details of the constructed geometry and presents with discussion the obtained results.

PHOENICS, CHAM's general-purpose fluid flow and heat transfer code is well suited to the modelling required; it has been successfully applied to similar problems in the past.

2. THE PHOENICS MODEL

2.1 Geometry and computational grid

The geometry of the cabinet was split into single units (parts) using Autocad Mechanical desktop then converted into STL file format. The single units that made up the geometry were then imported into PHOENICS environment using the Cad to



PHOENICS interface. The PHOENICS representation of the geometry is shown in the figures below.

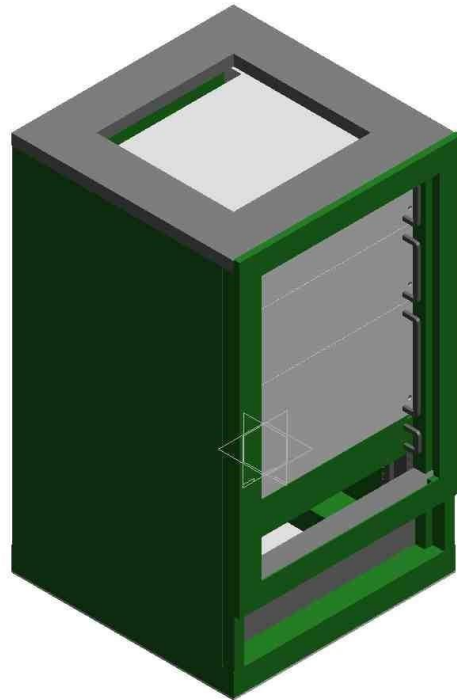


Fig.1 Full geometry of cabinet

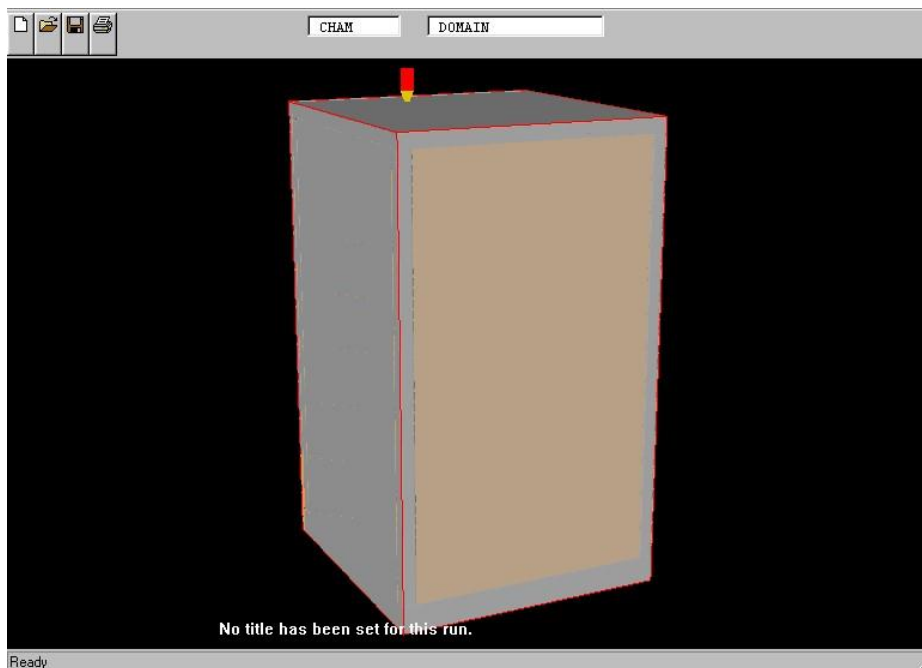




Fig.2 Front view of cabinet after import into PHOENICS VR using Cartesian coordinate system

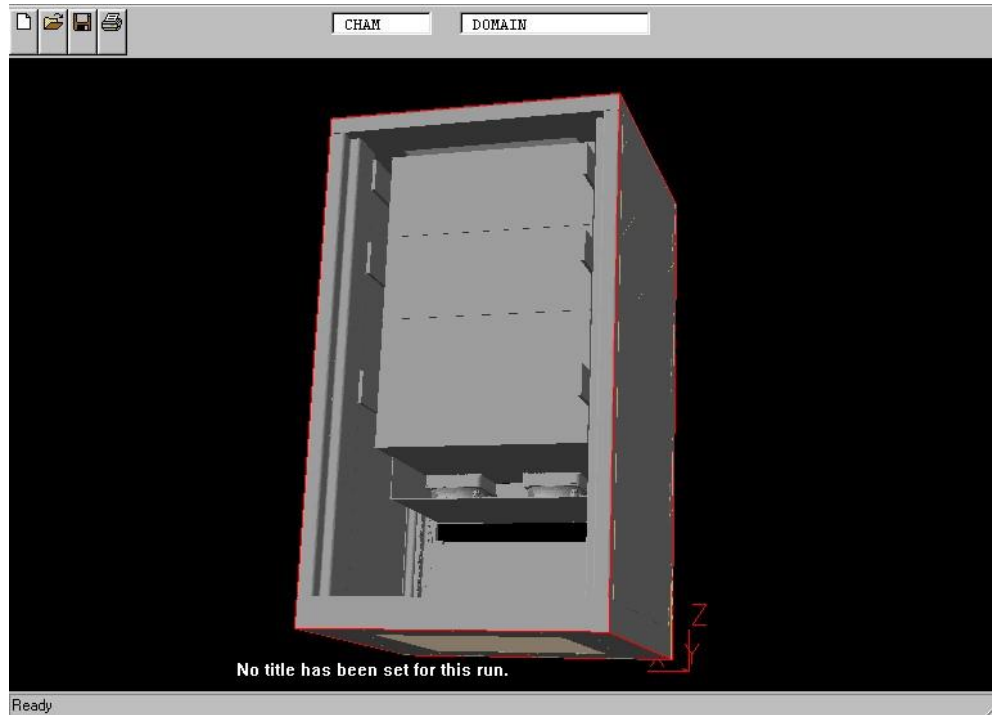


Fig.3 Back view of cabinet after import

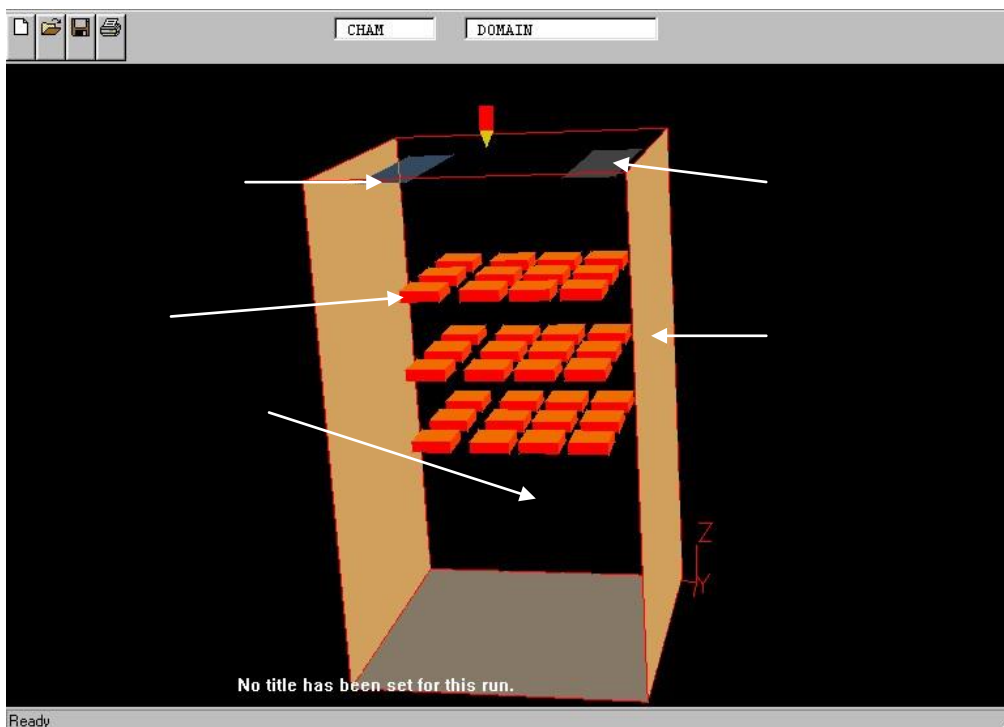




Fig.4 Internal objects after hiding the external geometry

A Cartesian grid of 35 * 57 * 50 cells was used to represent the geometry as shown below.

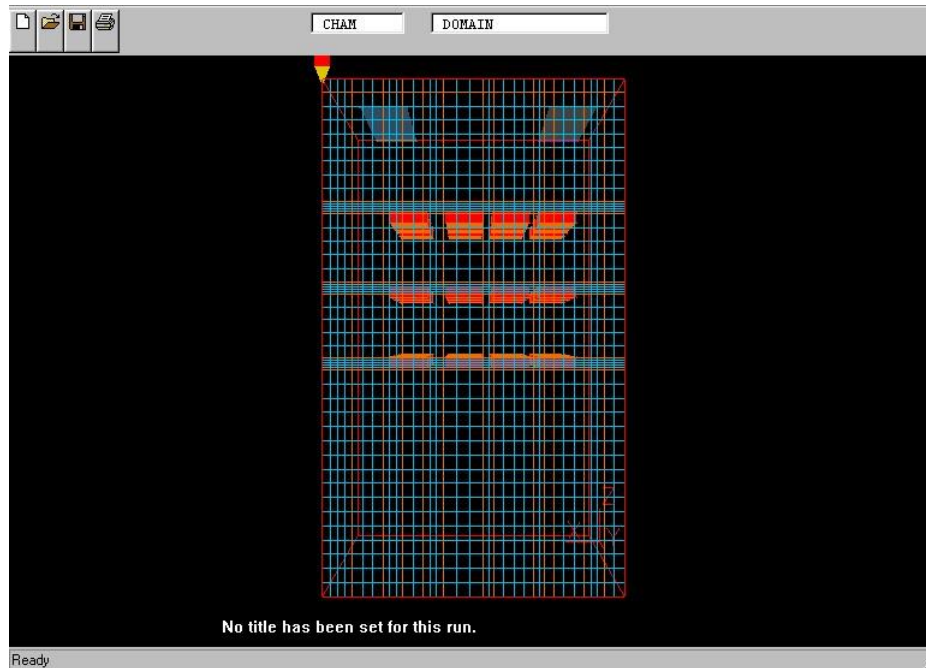


Fig.5 Grid used in XZ plane

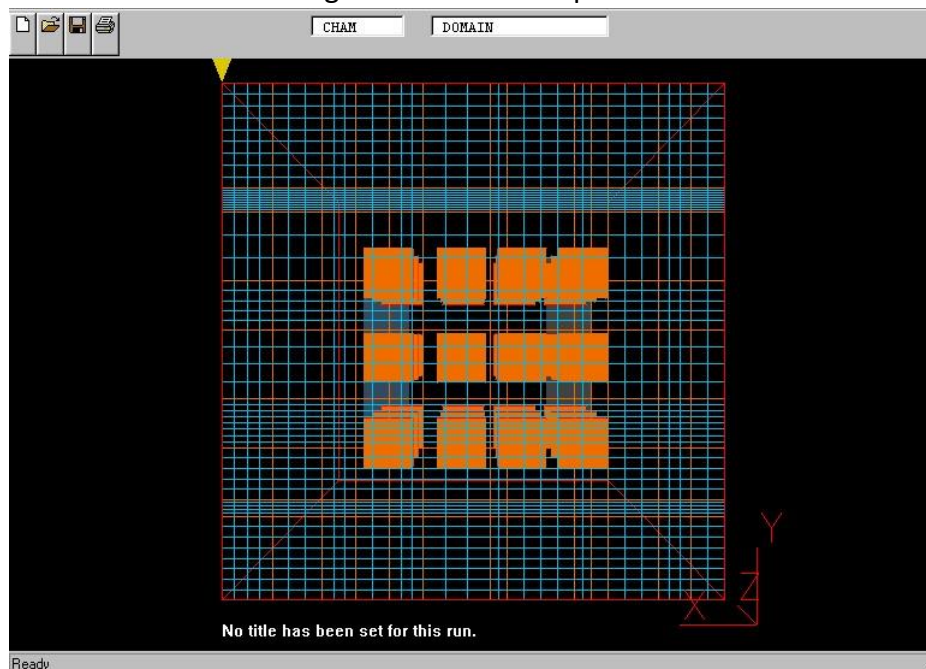


Fig.6 Grid in XY plane

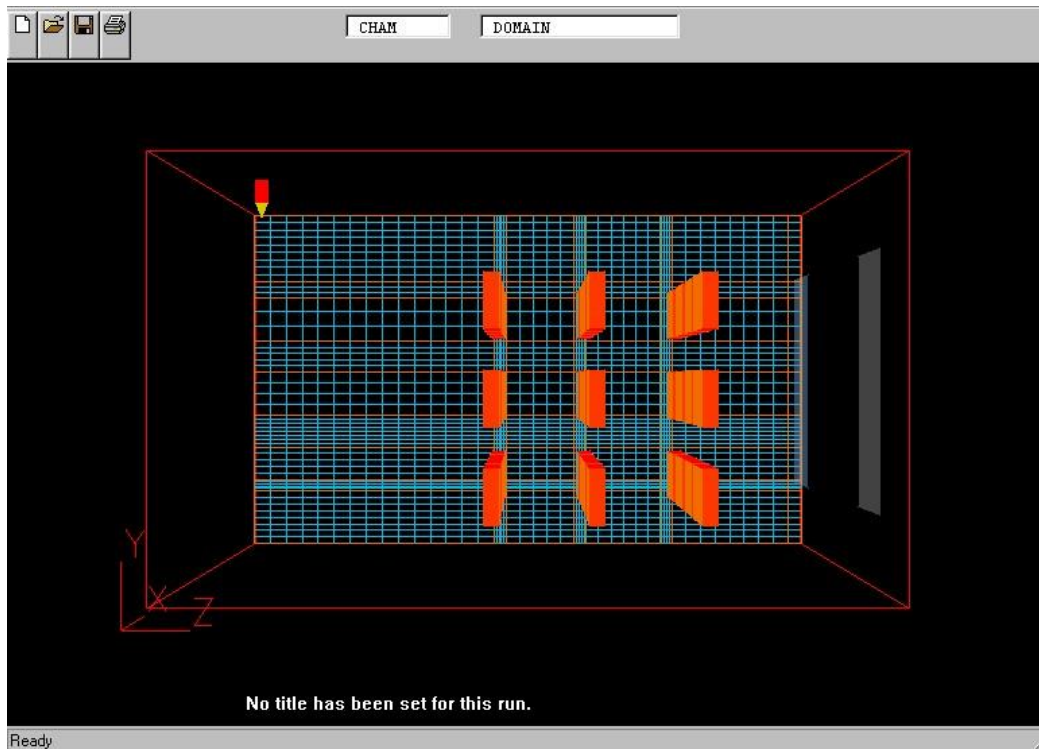


Fig.7 Grid in ZY plane

2.2 Boundary Conditions and Fluid Properties

The fluid was taken to be Air using ideal gas law: Gas constant = 287

Gamma = 1.4

Pressure = 1bar

CP = 1004 J/kgK

Density = 1.189 (kgm⁻³ Ideal gas) initially then the density of the gas (air) was calculated using:

$$\rho = P / R(\text{temp}+T)$$

Where P = pressure, temp = actual temperature, R = gas constant, T = 273.

The standard K- ϵ turbulence model was used to represent the effects of turbulent diffusion.

One inlet supplying forced convection air at a rate of 1.0 m/s was defined. The outlet was treated a fixed-pressure boundary zone (as per specifications).



2.3 Variables solved

For the three-dimensional, steady state, single phase simulation with conjugate heat transfer **PHOENICS** solved six equations, predicting three dimensional flow fields for the following dependent variables:

- X-direction velocity component (u).
- Y-direction velocity component (v).
- Z-direction velocity component (w).
- Pressure P.
- Temperature T.
- Turbulence KE and EP.

The values of the temperature and air velocity were stored and results are presented below.

3. REPRESENTATION OF THE RESULTS

3.1 Results

The main task of this demonstration was to show the capability of PHOENICS in predicting airflow characteristics inside the cabinet.

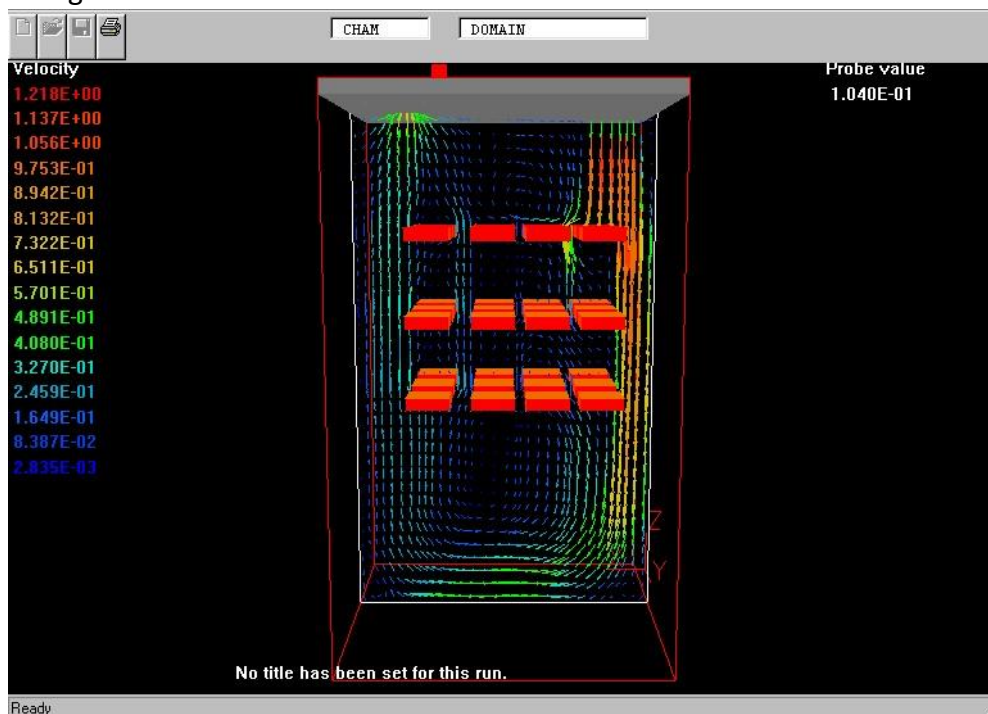


Fig.8 Velocity distribution

In the above Figure the airflow enters the domain from the top right hand side at a speed of 1 m/s & temperature of 39°C and leaves from the top left hand side. During this process heat exchange takes place between the fluid (air) and solid (heat sources) as shown in Fig.10.

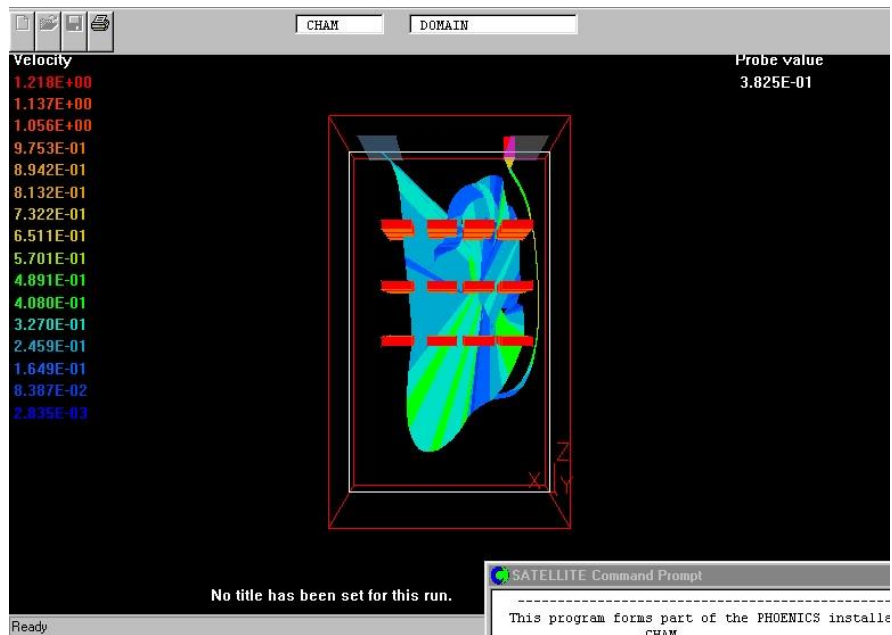


Fig.9 Snap shot of stream line

The above Figure shows a trace of a streamline (particular vector) from the point it enters the domain until it leaves. As can be seen, during this process the velocity vector circulates in the domain dissipating its momentum and gaining heat from the heat exchange between the solid and fluid.

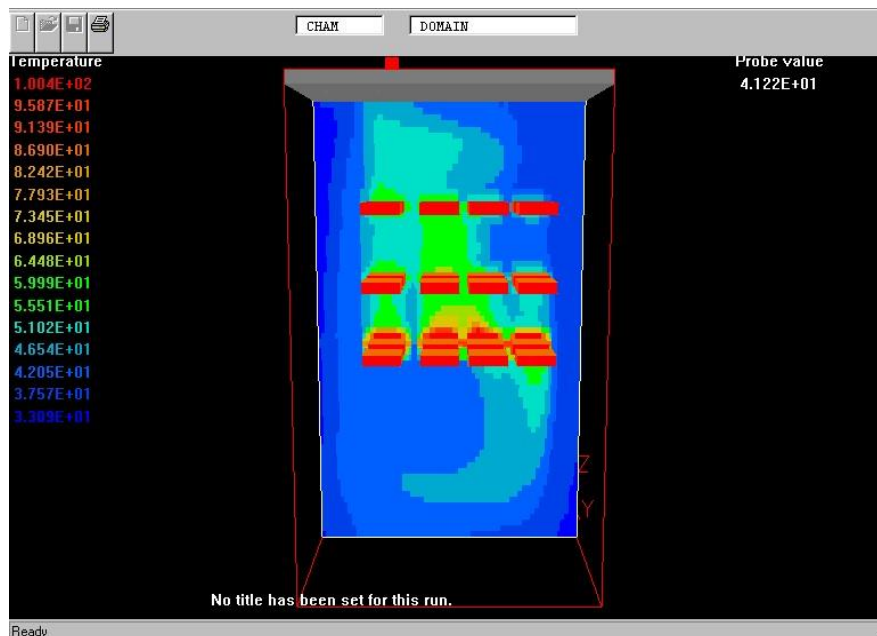


Fig.10 Heat dissipation from the solid to the fluid

This figure shows the temperature distribution as a result of applying a cooling load of temperature of 39 C and velocity of 1.0 m/s.



3.2 Conclusions from the study.

The results shown above show that the incoming air tends to keep circulating in all sorts of directions before it finally leaves through the outlet. This is very good as it indicates a successful definition of the inlet and outlet.

However it was lengthy procedure to convert and split the geometry into single objects, the setup of the case took approximately 45 minutes to set up and 2.5 hours to compute on PII/260.

The model developed here has many potential uses for further study, including the evaluation of different geometries, determining the sensitivity to operating parameters (e.g. velocity, temperature, etc.).

Contact: contact@arcofluidconsulting.com