Numerical Analysis of CPU with Heat Sink base of Copper Core using CFD

Guessab A., Aris A.

***Abstract*—**This research work presents a numerical simulation of the Computer Processing Unit (CPU) cooler heat sink with copper core and impingement air cooling. The Reynolds Averaged Navier-Stokes equations are discretized using the hybrid volume/difference finite scheme. The numerical simulations have been performed based on a commercial code ANSYS Fluent 14.0. Thermal characterization of heat sink under air-forced convection cooling condition is investigated. The thermal parameters such as velocity profiles, the static pressure distribution, the pressure drop and temperature (fluid/solid) distributions through the fins, the base heat sink and the heat sink body through the heat sink are presented and analyzed. The performance of the proposed model, which is tested through numerical computations is excellent compared to available results in literature.

***Keywords*—**CPU, RANS, Heat sink, ANSYS, VFM.

I. Introduction

In industry, thermal analysis are often ignored in the design process or performed too late. When design changes are limited and become too costly the numerical method used in this article, can be used to determine an estimation of the heat sink or component temperatures of products before a physical model has been made. A numerical model is normally used as a first order estimate. Since numerical methods for computational fluid dynamics (CFD) provide a qualitative and sometimes even quantitative prediction of fluid flow. Online heat sink [1].

CPU cooling is a critical aspect of a functioning computer system, and for this reason, forced-air cooling is a significant factor that should be determined at an early stage in system design. At certain point any electronic device can be irreversibly destroyed (typical maximum operating temperature for semiconductors is 125-175C at their junctions, capacitors 85-125C, wire insulation 105-200C. the thermal management and engineering whose task is to control 'T' of the product, is therefore an essential part of electronics design. Conventional electronics cooling normally using impinging jet with heat sink showed superiority in terms of unit price, weight and reliability. Therefore, the most common way to improve cooling leans on the use of air jet impact on a mini or micro channel heat sink. In this paper, the simulation of plate-fin heat sinks with confined impingement cooling in thermal-fluid characteristics is numerically investigated.

Here, our objective is focused on the impingement flow plate fin geometry. The main objective is to develop a simple model to predict thermal and hydraulic performances of plate fin heat sink for impingement air cooling.

A. Guessab, Laboratoire de Recherche en Technologie de Fabrication Mécanique (LaRTFM), Ecole Nationale Polytechnique d'Oran, Algeria, 31000., Oran (corresponding author to provide phone: 00213. 560706424; (e-mail: ahmed.guessab@enp-oran.dz).

A. Aris, Laboratoire de Recherche en Technologie de fabrication Mécanique (LaRTFM), Ecole Nationale Polytechnique d'Oran, Algeria, 31000. (e-mail: abdelkader.aris@enp-oran.dz).

II. Physical Model

The physical model of the considered study is illustrated in Fig. 1. This figure show the computational domain size and the components of the CPU Cooler. It is a 3-D model in dimensions of (X_{max} = 95 mm, X_{min} = - 95mm ; Y_{max} = 112,3mm, Y_{min} = 0,5mm and Z_{max} = 50mm, Z_{min} = - 50mm).

In this study five configurations (Table 3), a heat sink (Fig. 2) heated from the bottom with a power (Q) generated by the CPU, which is absorbed by the sink and released to the atmosphere through the fins topped by an axial fan that blows air to dissipate heat in the atmosphere with a constant flow rate.

The sink is made of aluminum. The power dissipation from the CPU cooler is set to 75W. The following assumptions are made to model the heat transfer and fluid flow in the heat sink (3D fluid flow and heat transfer): (1) steady state, (2) turbulent, (3) constant fluid and solid properties, (4) negligible viscous dissipation, (5) negligible radiation heat transfer.

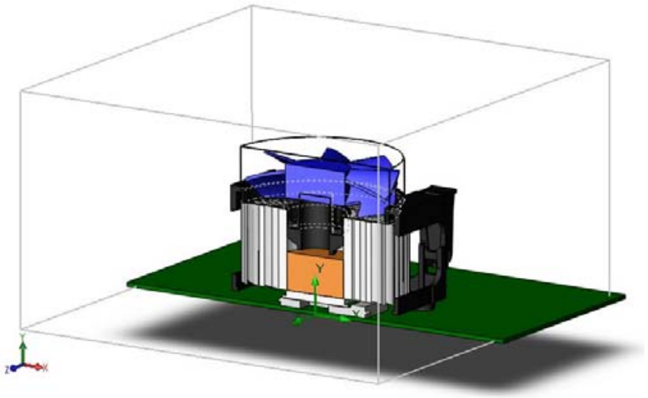
The dimensions of the heat sink considered in this work are shows in Fig. 2. The governing equations of continuity, momentum and energy are solved numerically using the hybrid finite volume/element with boundary conditions as follows (Table 1). The default turbulence model of all calculations is the Mixing Length/Algebraic Turbulence Model (zero equation). Algebraic Turbulence Model is economical and accurate for most electronics cooling applications.

In this paper, the thermal resistance of the heat sink is calculated by:

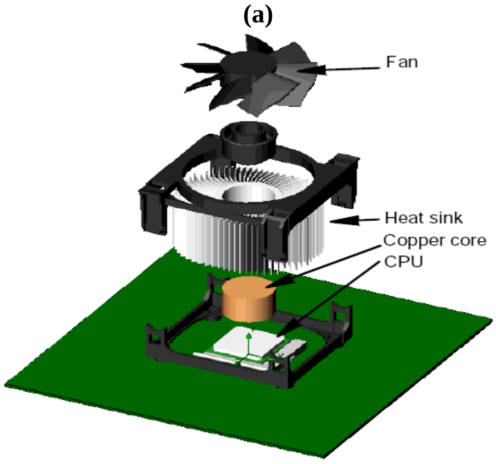
$$R_{Total} = \left(T_h - T_l\right) / \phi = R_{Convection} + R_{Conduction} \tag{1}$$

R_{Total}, $R_{convection}$ and $R_{conduction}$ represent the total thermal resistance, the convection thermal resistance and the conduction thermal resistance respectively.

The iterations were stopped when the residuals for the continuity, momentum equations were 1% of the characteristic flow rate and were 0.1% for the energy, Thus, the resulting solution is convergent.



(a)



(b)

Fig.1 Schematic diagram showing the physical model: (a) Overall drawing, (b) exploded drawing.

Parameter	Value	description
T_∞	28°C	Air temperature
Q	75W	Power of heat source
N	4400 rpm	Angular velocity
T_S	28°C	Solid temperature
A_{CPU}	968mm^2	Surface of CPU

Table1 : Boundary conditions operating parameters of the model.

Property	Aluminium	Copper
Density, Kg/m^3	2719	8978
C_P, J/Kg.K	871	381
λ, W/m.K	201.4	387.6

Table 2 : Heat sink material and its properties.

III. Mathematical Formulation

In this study, the three-dimensional time-averaged continuity, Navier-Stokes and energy equations are written as follows.

$$\frac{\partial(\rho u_j)}{\partial x_i}=0 \quad (1)$$

$$\frac{\partial(\rho u_j u_i)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \quad (2)$$

$$\frac{\partial(\rho u_j H)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(k_{eff}\frac{\partial H}{\partial x_i}\right) \quad (3)$$

where the effective viscosity μ_{eff} and the effective thermal diffusivity k_{eff} are defines as

$$\mu_{eff}=\mu+\mu_t \quad \text{and} \quad k_{eff}=\frac{k}{Pr}+\frac{k_t}{Pr_t} \quad (4),$$

(5)

For the solid :

$$\lambda_S\nabla^2 T=0 \quad (6)$$

IV. Method of Solution

It is noted that the default turbulence model of all calculations is the Mixing Length/Algebraic Turbulence Model (zero equation) are available in Ansys Fluent. The ANSYS FLUENT preprocessor GAMBIT was used to creat the mesh. The computational domain is a rectangular enclosure with a height of 190 mm, a depth of 0.1118 m and a width of 190 mm. The specifications of the heat sink considered in this surdy, are presented in Table 3. Both types of heat sinks are assumed to be made of aluminum and the material properties are taken accordingly. In this configuration, a heat sink is heated from the bottom with a power (Q) generated by the CPU, which is absorbed by the sink and released to the atmosphere through the fins topped by an axial fan that blows air to dissipate heat in the atmospher with a constant flow rate V(m^3/s). The same preprocessing software GAMBIT has been used to model both the fins. Conjugate convection has been considered and so nodes have been placed inside the fins too. The heat sink fan is a 60 mm diameter circular internal fan with a specified fan performance curve. The exhaust fan is 85 mm×85 mm in size. ANSYS Fluent assumes that fluid exits the fan in the direction normal to the plane of the fan. This skews the flow direction in the θ direction, i.e., the direction of blade revolution. Swirl magnitude is defined by

$$u_\theta(r)=u_Z(r)\left(\frac{r}{R}\right)S \quad (7)$$

where $u_\theta(r)$ is the velocity in the direction of revolution, $u_z(r)$ is the velocity in the direction normal to the fan, *r* is the radial coordinate, *R* is the outer radius of the fan, and *S* is the swirl

magnitude. Instead of using a specified swirl magnitude, ANSYS Fluent can also allow the swirl factor to change as a function of the operating point on the fan curve.
This is achieved by specifying the RPM (rotation per minute) of the fan. The swirl magnitude (the ratio of the tangential velocity to the axial velocity) can then change as the fan operating point changes on the fan curve. Fan RPM is defined by

$$u_{\theta}(r)=\left[(\mathrm{RMP})\times\frac{2\pi}{60}\times r\right]\frac{1}{20} \tag{8}$$

The segregated solver is the solution algorithm used in this study, which solves the governing equations of mass, momentum and energy sequentially (i.e. segregated from one another). A second order upwind scheme is employed for the discretization of all the variables and the SIMPLE scheme has been used for the pressure-velocity coupling. The numerical solution is considered to be converged when the residual for continuity, turbulence quantities and velocity terms are within 1×10^{-3}, and the residual of the energy equation for the domain is within 1×10^{-6}.

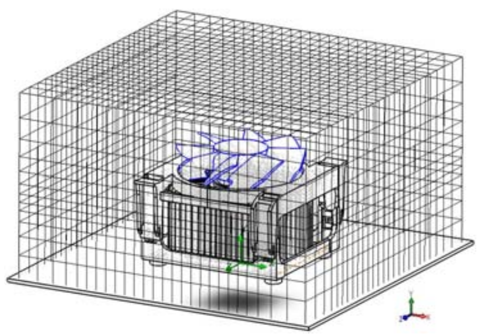


Fig. 2 3D-CFD domain meshing

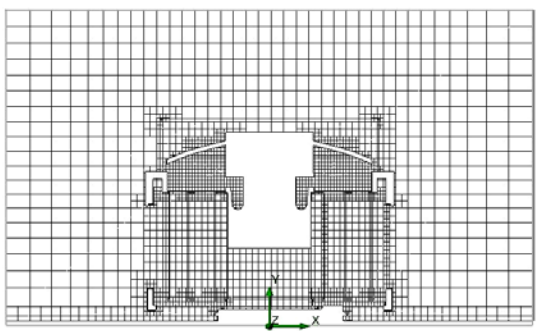


Fig. 4 2D-CFD domain meshing

V. Results and discussions

This work first conducts a parametric sudy on the effects of jet Reynolds numbers, vertical spacing between the nozzle and the heat sink, width and length of fins, and thikness of the base plate of the heat sink. Next, a chosen heat sink with a suitable spacing is installed in a personal computer to evaluate the thermal performance under various operating conditions.

The specifications of the heat sink considered in this study are listed in Table 3.

Fin N°	Fin array $N_x\times N_y$	Fin Number	D_f (mm)	H_s (mm)
1	16×10	42	2	27.573
2	16×12	52	2.27	32.476
3	30×16	65	2.64	34.951
4	40×10	72	3	37.524
5	16×30	82	3.37	42.476

Table3 : The specifications of the heat sink considered in this surdy.

Figure 6a illustrates the results of the thermal resistance of the heat sink based on different grids. It can be seen clearly that the thermal resistance is almost unchanged within the range of grids from 120000 to 195862.

Figure 5b depicts the variations of the total thermal resistance of the heat sink vs. Size of heat sinks subjected to different jet Reynolds numbers. It can be seen clearly that the thermal resistance decreases as Reynolds number increeases due to the enhancement of cooling. However, such an increases in cooling with increasing Reynolds number diminishes gradually.

Figure 5b also indicates that the heat sink of 65 fins displys the lowest thermal resistance for all jet Reynolds numbers considered in this work. The reason for such phenomena can be explained as follows. When the number of the fins increases beyond on optimal value, it can result in a decreases for the width of flow channel and hence a larger pressure drop.

This yields shallow penetration for the jet into the space between fins, and causes insufficient cooling and hence a higher thermal resistance. Based on 65 fins, Re = 15000, fin width = 2.64 mm, the effect of the fin height is shown in Figure 6. It indicates that increasing the fin height can decrease the thermal resistanc ; however, such a decrease in the tharmal resistance tends to die down. This is because indreasing the fin height can increase the heat transfer area whereas it is more difficult for the flow to penetrate into the inside surface of the heat sink. The effect of the fin width is shown in Figure 7 for 65 fins, Re = 15000, fin height = 32.5mm. A apparently, an increases in the fin width can cause a decrease in the thermal resistance.

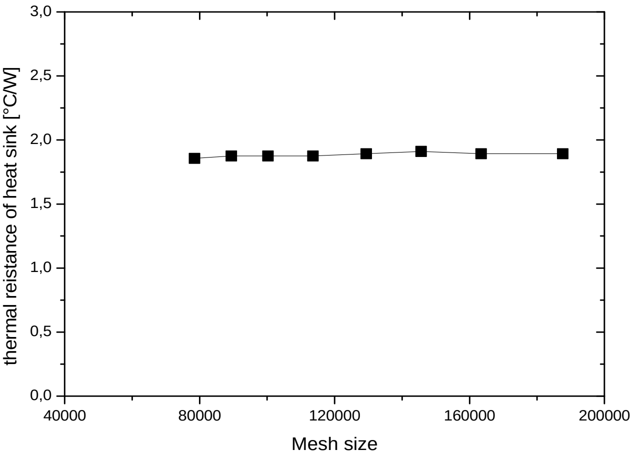
A minimum for the thermal resistance appears. This is due the fact that when the fin width increases such that the fluid channel reduces below the optimum, the jet cannot penetrate into the space between fins freely. Befor we discuss the results, it is essential to examine the accuracy of the numerical scheme used in the work. The only way to establish grid independent solutions is to setup a model with a finer mesh and analyze it to see if there are major differences in scalar quantities and vectors. An additional test case is prepared using 78945 cells (Fig. 8a).

The results are compared with the default 195862 cell model (Fig. 8b). Figure 5a depicts the variations of the total thermal resistance of the heat sink vs. size of the heat sinks subjected to different flow Reynolds numbers. It can be seen clearly that the thermal resistance decreases as Reynolds numbers increases due to enhancement of cooling. Figure 8b also indicates that the heat sink of 65 fins displays the lowest thermal resistance for all the flow reynolds numbers considered in this work.

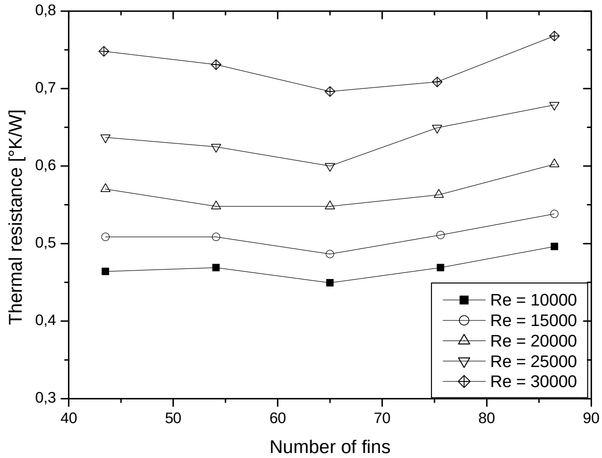
The mesh density increase mostly concentrated within the nonconformal mesh around the heat sink. Figure 8 illustrates that the temperature distributions are similar. This shows that 195862 cells are enough for the models to be grid independent. The density distribution of the mesh is concentrated around the CPU heat sink. The value of the maximum temperature of the center of the CPU cover is 57,48°C. Now we can calculate the thermal characterization parameter of the heat sink:

$$R_{Total} = (T_h - T_l)/\phi = R_{Convection} + R_{Conduction}$$
$$= (57.48 - 38)/75 = 0.26°C/W \quad (9)$$

The second most important characteristic of the CPU Cooler is the velocity of the flow above PCB. We can assess the value of this parameter as well as the distribution of the temperature by looking at the cut plots made in the Front plane (Fig. 9). As the cooling air from the fan does not provide a uniform flow pattern, there are regions in the heat sink wich show a remarkable temperature field. In general, if high temperature gradients are oberved in heat sink, the conduction effect is subject to improvement. In Figure 9, the distribution of the surface temperatures is shown. Looking at the flow vector field in a center cross section, above left, the thermal simulation reveals a zone below the fan motor with no flow condition can be identified. This behavior is seen for all axial fans used in this type of application. Thermal imaging shows the highest temperature directly at the contact area between CPU and heat sink. The heat is transported inside the heat sink aluminium material via conduction and from the surfaces to the air via convective heat transfer (Fig. 10).



(a)



(b)

Fig. 5 (a) Thermal resistance of the heat sink based on various grids for Re = 15520, (b) Thermal resistance vs. number of fins at various Reynolds numbers.

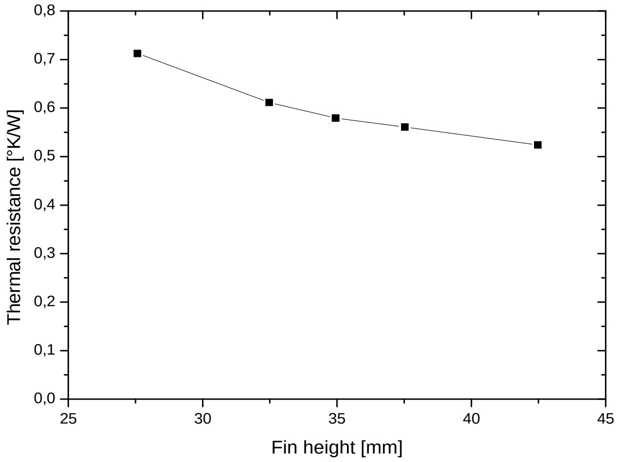


Fig. 6 The dependence of thermal resistance on the fin height for 65 fins, z/D = 8, Re = 15000, and fin width = 2.64 mm.

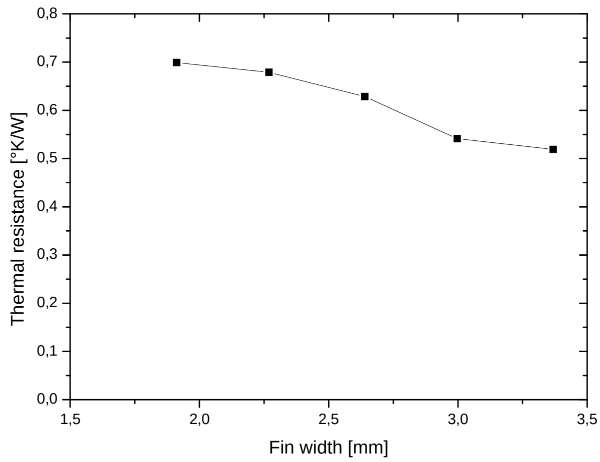
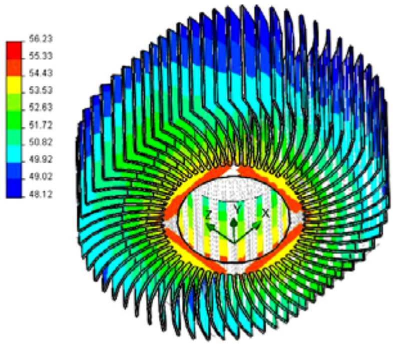
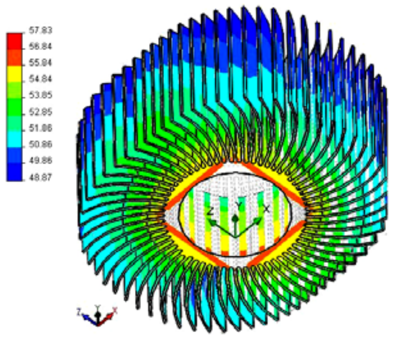


Fig. 7 The dependence of thermal resistance on the fin width for 65 fins, z/D = 8, Re = 15000, and fin height = 32. 5 mm.



(a)



(b)

Fig .8 Grid independent solution: Temperature distributions through the heat sink, (3D).

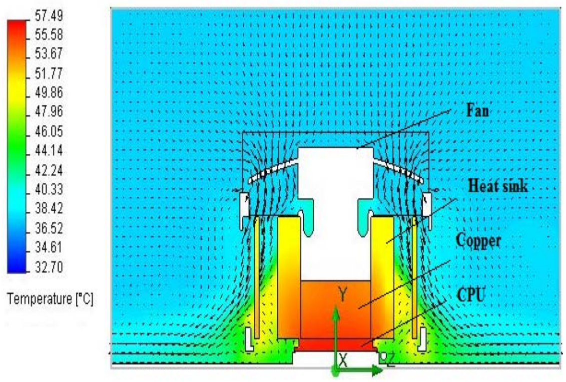


Fig .9 Temperature field and velocity vectors distribution

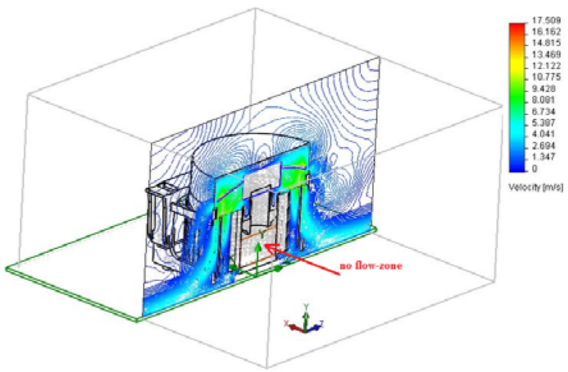


Fig. 10 Velocity vectors distribution

CONCLUSION

The important conclusions that can be drawn from the present study are given below:

(1) In this paper, the heat sink with copper core were investigated using ANSYS Fluent and the results were acceptable.

(2) The CPU case temperature has a linear variation with power dissipation. The heat sink temperature difference results it shows the good correlation

APPENDIX

CFD	Computational Fluid Dynamics
CPU	Computer Processing Unit
L	Characteristic length, m.
Gr	Grashof Number.
Nu	Nusselt Number.
Pr	Prandtl Number.
R	Ideal gas constant.
R	Thermal resistance, (m2・K)/W.
Ra	Rayleigh Number.
Re	Reynolds Number.
μ_{eff}	Effective viscosity
k_{eff}	Effective thermal diffusivity
T	Temperature

REFERENCES

[1] http://heatsinkcalculator.com

[2] Channamallikarjun, Thermal Analysis of CPU with variable Base plate Heat-sink using CFD, International Journal of Research in Engineering and Technology. Vol.3, Special Issue: 03, 2014.

[3] R.Mohan and Dr.P.Govindarajan, Thermal Analysis of CPU with variable Heat Sink Base Plate Thickness using CFD. International Journal of the Computer, the Internet and Management, Vol. 18 No.1, pp 27-36, 2010.

[4] Zhi-ping Xu, Qian Zhu, The Simulation and Optimization of the CPU Heat Sink for a New Type of Graphite. Journal of Environmental Engineering and Technology Vol. 1, No. 3, 2012.

[5] Emre Öztürk and Ilker Tari, Forced Air Cooling of CPUs with Heat Sinks: a Numerical Study, IEEE Transactions on Components and Packaging Technologies, Vol. 31, No. 3, pp. 650-660, 2008.

[6] [6] Ansys Fluent 14.0, help manual, http://www.ansys.com

Guessab Ahmed was born in Algeria, Planteurs City, Oran, in 1974. He received his Bachelor degree in mathematics from Ibn Badiss School, Oran. Engineer degree in Mechanical Engineering from Oran University Algeria (USTOMB) in 2002, and he has Magister Degrees, in energetic from mechanical institute Oran University (USTOMB) in 2004. He was a teacher of some disciplines in National Polytechnic School from 2005 to present day. He focuses his research interests on the turbulent flow, models of turbulent combustion and combustion CFD simulation.

Aris Abdelkader is an Assistant Professor of Mechanical Engineering at the National Polytechnic School of Oran, Algeria. He received his Ph.D in Mechanical Engineering from Oran University, USTO.MB (Algeria) in 2006. His researches focus on numerical and experimental combustion phenomenon (internal combustion engine, burners and furnaces), heat transfer and renewable energy. He is responsible for the team's energy specialty in engineering school (ENP.Oran)