Journal of Engineering and Sustainable Development

Vol. 22, No.05, September 2018
ISSN 2520-0917
https://doi.org/10.31272/jeasd.2018.5.12

Numerical Solution of Distributions For A Fluid Velocity, Pressure And Temperature Along Flow Path For A Non-Conventional Machining Method

*Dr. Basima Salman Khalaf

Lec., Mechanical Engineering Department, Mustansiriyah University, Baghdad, Iraq.

Abstract: This work deals with a Non-Conventional machining cell consist of copper tool of dimensions (30 mm width and 50 mm length and 7 mm thickness) ,En8 mild steel work piece of dimensions (30 mm width and 50 mm length and 8 mm thickness) and fluid of (10 % NaCl w/w). The flow into machining cell was fully developed turbulent flow. For the numerical solution, ANSYS FLUENT software used to solve the basics equations. The inlet operating parameters for this cell were selected within the range of industrially realistic machining circumstances(supply voltage 12 to 18 volt, tool feed rate 0.35 to 1.65 mm/min, fluid flow rate 5 to 30 lit/min, fluid temperature 40 °C and fluid back pressures 0 to 6 bar). The distributions of fluid velocity, static pressure and temperature along flow path were obtained. The results show the increasing of fluid temperature toward the outlet flow.

Keywords: *Numerical Solution; Distributions; Fluid Flow Rate; Fluid Pressure; Fluid Temperature.*

الحل العددي لتوزيع سرعة، ضغط ودرجة حرارة مائع على طول مسار جريانه بطريقة تشغيل غير تقليدية

الخلاصة: يتعامل هذا البحث مع خلية تشغيل غير تقليدية تتألف من عدة نحاسية ذات ابعاد (30 mm عرض ، 50mm طول و 7mm سمك) معدن المشغولة (En8 mild steel) ذات ابعاد (30 mm عرض ، 50mm طول و mm8 سمك) والمائع(10% NaCl w/w).تم افتراض الجريان داخل الخلية اضطرابي تام الاستحداث. استخدم برنامج ANSYS FLUENT لحل المعادلات الاساسية. ان قيم العوامل المؤثرة اخذت ضمن مدى عملي لتشغيل منظومة تشغيل غير تقليدية (فولتية التشغيل 12 الى 18 فولت ، معدل تقدم العدة 0.35 الى 1.65 mm/min ، معدل جريان المائع 5 الى 30 lit/min ، درجة حرارة المائع 40 °C والضغط الخلفي للمائع 0 الى 6 bar) وقد تم الحصول على توزيع السرعة، الضغط الاستاتيكي ودرجة الحرارة للمائع على طول الجريان. اظهرت النتائج زيادة درجة حرارة المائع باتجاه خروجه من خلية التشغيل.

1. Introduction

Ansys Fluent provides full modeling capabilities for a varied range of incompressible and compressible, laminar and turbulent fluid flow problems. In ANSYS FLUENT, a wide range of mathematical models for transference phenomena (like heat transfer and chemical reactions) is shared with the ability to model difficult

**Corresponding Author* drbasima2016@gmail.com

geometries. The finite element method is a numerical method which can be used for the precise solution of complex engineering problems. One of the main reasons for the popularity of the method in different fields of engineering is that once a universal computer program is written, it can be used for the solution of any problem only by changing the input data [1].

J. Kozak [2] presents models (physical and mathematical) for the simulation used Computer-Aided Engineering System for a non-conventional machining process. The application of this system results compared with the computer simulation. T. Paczkowski [3] analyses theoretically a non-conventional machining with curvilinear profile tool which vibrating in two directions. He used partial differential equations of the basic equations to describe the inter gap nature. The flow through gap was mixture of fluid and gas. A. Mishra, et al [4] studied the fluid flow pattern to accurately predict the machining variable distribution. They consider steady state with turbulence for inlet pressures. They conclude that the velocity of flow decreases at the work piece inlet and it increases at the work piece outlet. L. Zhiyong and N. Zongwei [5] prepared a model using a finite element numerical method of tool design. They work to minimize the convergence of finite element method and increase the accuracy in tool design, through tool shape iterated the tool shape to eliminate the design errors in computational process.

2. Mathematical Modeling

2.1. Assumptions

1- The heat generation at gap between work piece and tool only.
2- The fluid flow rate flow though uniform channel.
3- The height of work piece or tool is much larger much bigger than the gap height.
4- The voltage is kept constant throughout the analysis for each case.
5- No change in properties of work piece or tool materials while it changed for the fluid.
6- Work piece and tool geometry were consider isotropic and homogenous.
7- The joule's heating effect can be applied into gap between work piece and tool.
8- The fluid flow is considered as a fully developed turbulent flow.

2.2 *Governing Equations*

The fluid motion governed by the Continuity, momentum and energy equations [6-8],then:-

a- Equation of Continuity:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{\upsilon}_m) = S_m \tag{1}$$

Where, $\vec{\upsilon}_m$ is the mass-averaged velocity and ρ_m is the mixture density.

$$\upsilon_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \upsilon_k}{\rho_m} \tag{2}$$

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{3}$$

Equation (1) is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The right hand term is the source which is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

b- Momentum Equation:

Conservation of momentum in an inertial (non-accelerating) reference frame is described by:

$$\frac{\partial(\rho_m \vec{\upsilon}_m)}{\partial t} + \nabla \cdot (\rho_m \vec{\upsilon}_m \vec{\upsilon}_m) = -\nabla P + \nabla \cdot [\mu_m (\nabla \vec{\upsilon}_m + \nabla \vec{\upsilon}_m^{\,T})] + \rho_m \vec{g} + \vec{F} + \nabla \cdot (\sum_{k=1}^{n} \alpha_k \rho_k \vec{\upsilon}_{dr,k} \vec{\upsilon}_{dr,k}) \tag{4}$$

Where P is the static pressure, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. Also contains other model-dependent source terms such as porous-media and user-defined sources. Also, n is the number of phases, μ_m is the mixture viscosity and $\vec{\nu}_{dr,k}$ is the phase (k) velocity

b- Equation of energy

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}(\alpha_k \rho_k E_k) + \nabla \cdot \sum_{k=1}^{n}(\alpha_k \upsilon_k (\rho_k E_k + P)) = \nabla \cdot (K_{eff} \nabla T) + S_E \tag{5}$$

Where

$K_{eff} = \sum \alpha_k (k_k + k_t)$ is the effective conductivity.

k_t is the fluid turbulent thermal conductivity.

The first term of the right hand side for the “(5)” showed the heat transfer and (SE) deals with volumetric heat generation.

d- Energy of the turbulence kinetic

This energy and its dissipation rate obtained from the solving of this equation:

$$\frac{\partial}{\partial t}(\rho K)\frac{\partial}{\partial x_i}(\rho K u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_i}{\sigma_k})\frac{\partial K}{\partial x_j}]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{6}$$

And

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_i}{\sigma_\phi})\frac{\partial\varepsilon}{\partial x_j}]+C_{1\varepsilon}\frac{\varepsilon}{K}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{K}+S_\varepsilon \tag{7}$$

The kinetic energy generation (G_k), Y_M is the fluctuating in turbulence to rate of dissipation and G_b is the kinetic energy generation.

σ_k and σ_ε are the Prandtl numbers for k and ε respectively,

$C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants,

S_k and S_ε are user-defined source terms.

2.3 Material

Tool material is copper and the material of work piece is En8 mild steel where the fluid is sodium chloride solution. The parameters used are inlet flow velocity, electrolyte viscosity and density and the physical properties of the materials. The gap heat generated was dependent on the density of current and fluid temperature. The heat transferred from fluid to the wall by conduction. “Fig. 1” shows the computational fluid dynamics (CFD) model analysis.

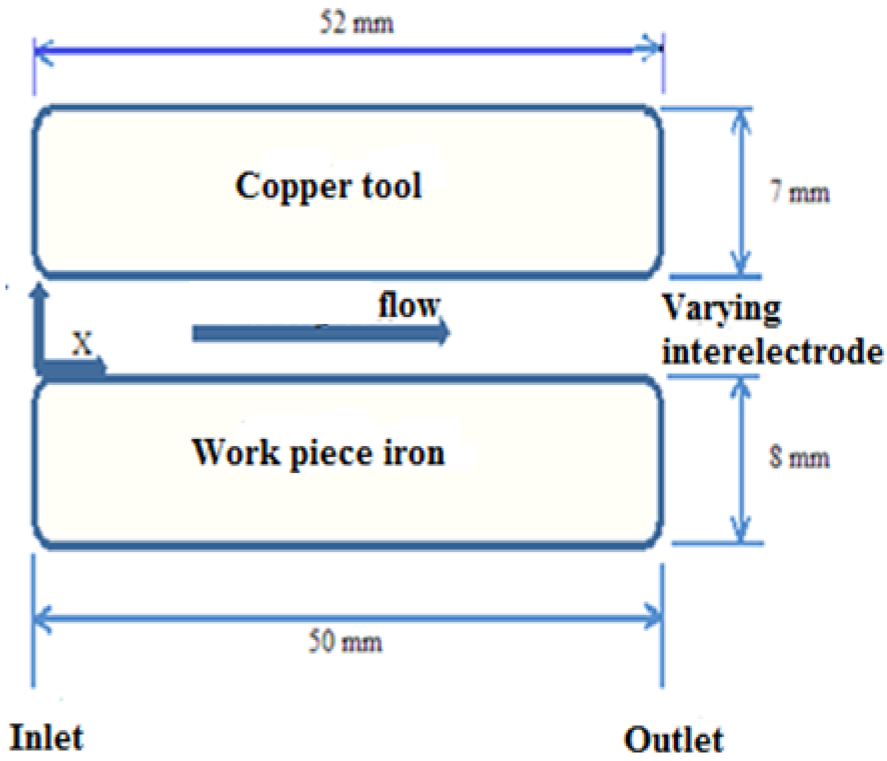

Figure 1. Physical model for CFD analysis

2.4 Type of Element

Firstly, the CFD model was prepared and meshed and generated for the element shown in the "Table 1". The element used in FLUENT analysis.

Table 1. The element used in FLUENT analysis

Element	Element Type
Solid	Iron
Solid	Copper
Fluid	Brine

2.5 Properties of material and Boundary Conditions

The Properties of Material and boundary conditions were presented in "Table 2". The material properties for copper, iron and electrolyte used in the analysis.

The wall heat transfer conduction coefficient work piece and tool= 1000 W/m^2 K .

Table 2. The material properties for copper, iron and fluid used in the analysis

The Property of Material	The Fluid	Copper Material	Iron Material
Density (Kg/m^3)	1070	8940	7860
Thermal Conductivity (W/m/K)	0.6	401	80
Specific Heat (J/Kg/K)	3760	390	460
Viscosity (Kg/m/s)	0.001	-	-

The inlet fluid velocity = (5 –140 m/s). The electrolyte inlet temperature = (40 $^{\circ}$C).

2.6 Condition of Electrical Boundary

- The voltages are in the range (12-18 volt) and the electrical currents are between (82-367 Amp).
- The pressure outlet boundary conditions (0-7 bar).
- UDF (User Defined Function) is introduce the heat generated.

The distributions of temperature, velocity and pressure were obtained after the boundary conditions applied. As shown in "Fig. 2", four sections were considered, three of them are perpendicular to x-axis while the fourth section stated parallel to x- axis. For these sections, the distribution of velocity, pressure and temperature will be shown.

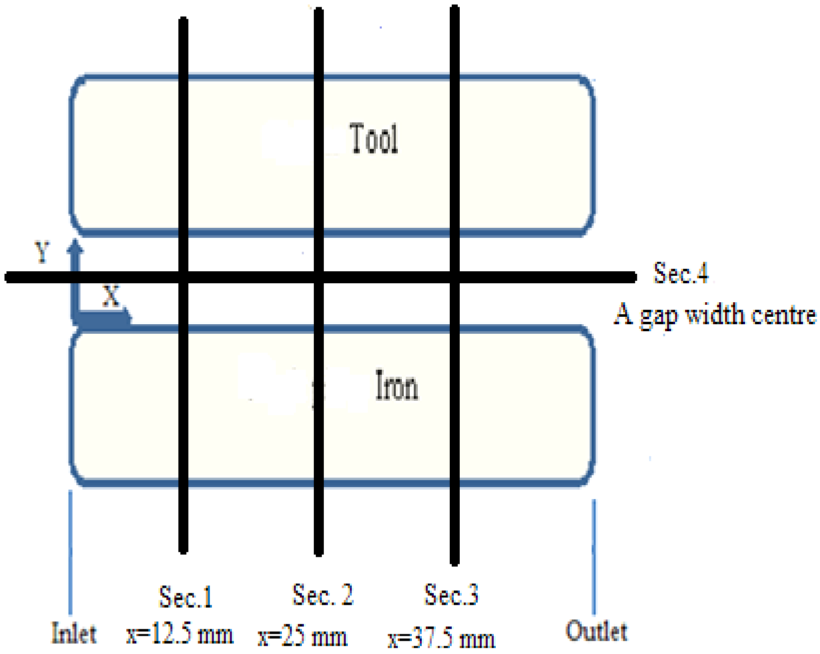

Figure 2. Sections of distributions locations

3. Simulation and Results

Velocity, pressure and temperature profiles were obtained by ANSIS FLUENT software for all test input values, these input values are supply voltages, machining currents, tool feed rates, fluid inlet flow rates, inlet temperatures and pressures inlet and outlet. “Table 3” shows the selected experimental input data test for simulation in this analysis.

Table 2. The material properties for copper, iron and fluid used in the analysis

Voltage (volt)	Current (Amp)	Tool Feed rate (mm/mi)	Fluid Inlet Flow rate (lit/min)	Fluid Inlet Temperature (° C)	Fluid Inlet Pressure (bar)
12	82	0.35	20	40	1.75

“Figs. 3 to 12” show the results for selected case of velocity, pressure and temperature to simulate the experimental results. The relations between flow path and the velocity, temperature and pressure showed in this analysis. These plots include graphical solution representations which is dividing to four sections. These figures give a good agreement simulation with experimental results for outlet values of velocity, pressure and temperature of the fluid along flow path. Also, the graphical solution with four sections gives a useful informations regarding distribution of fluid velocity in x-axis along a certain gap width with (x=12.5, 25 and 37.5 mm), and these locations will cover flow path, at the same time, the same observations could be applicable for electrolyte temperature. Fluid temperature distribution with section (4) which lies at the center of gap width, gives the variation of temperature along flow path.

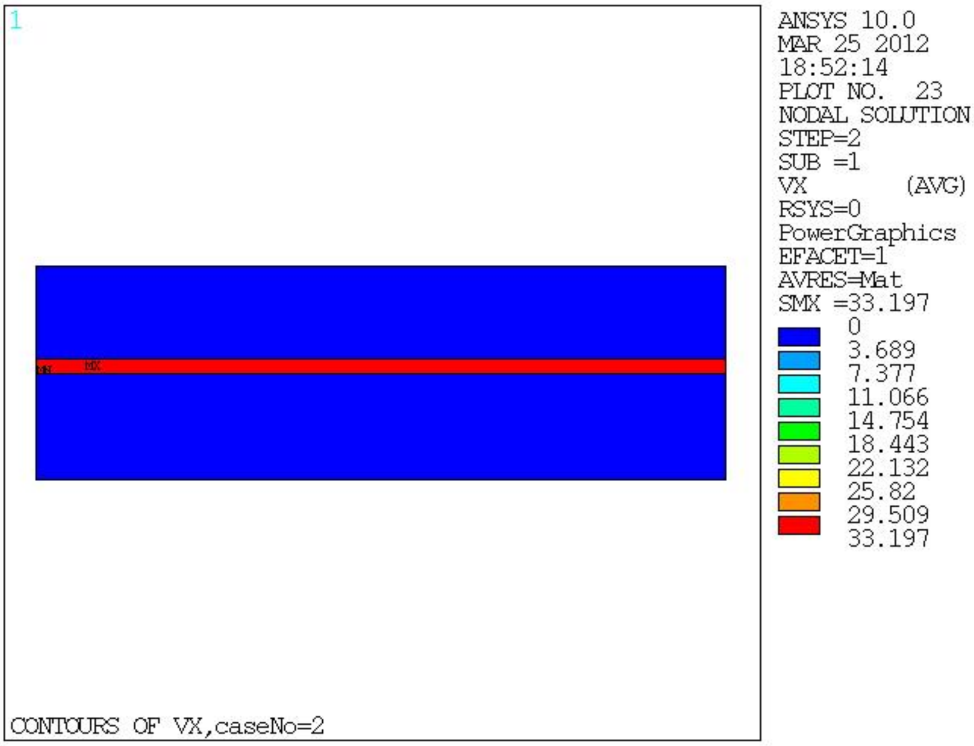

Figure 3. Simulation of fluid velocity

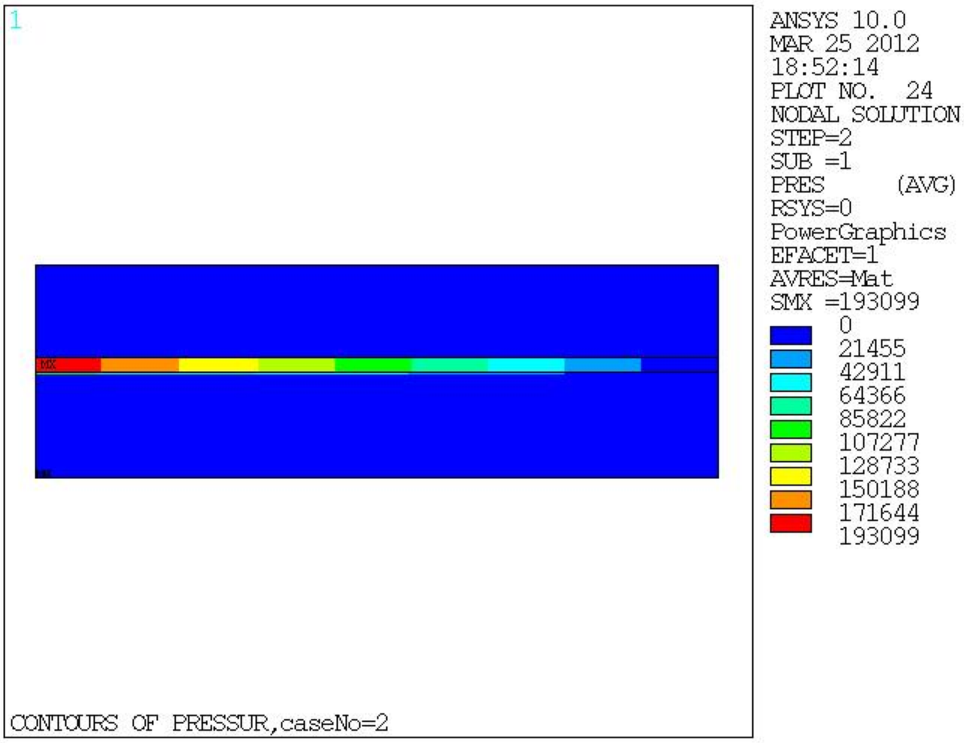

Figure 4. Simulation of fluid pressure

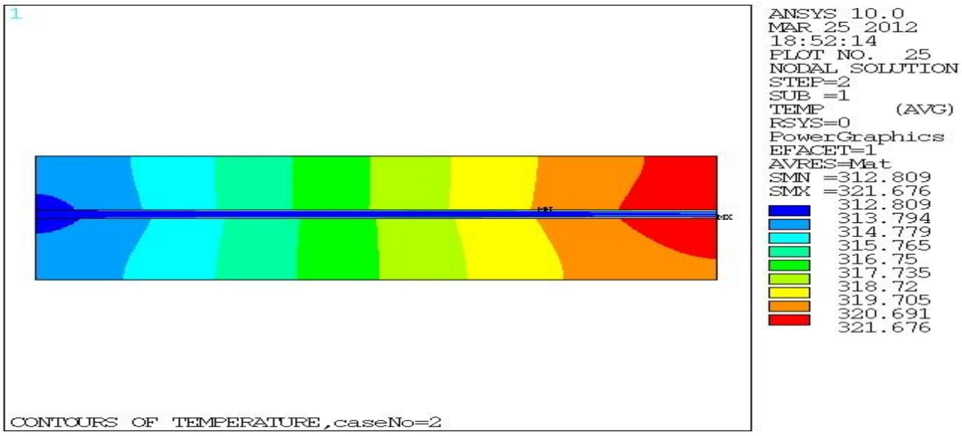

Figure 5. Simulation of fluid temperature

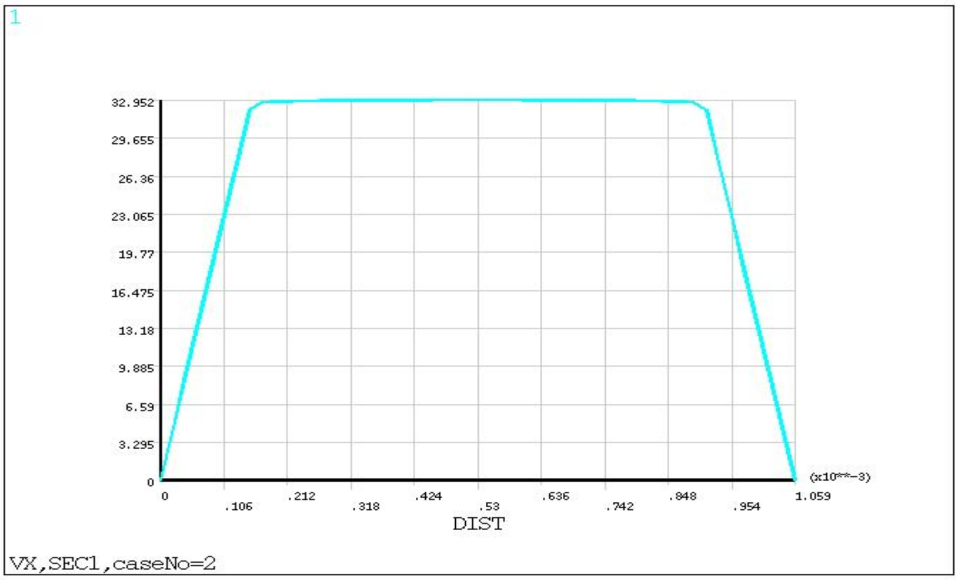

Figure 6. Graphical solution of fluid velocity at section (1)

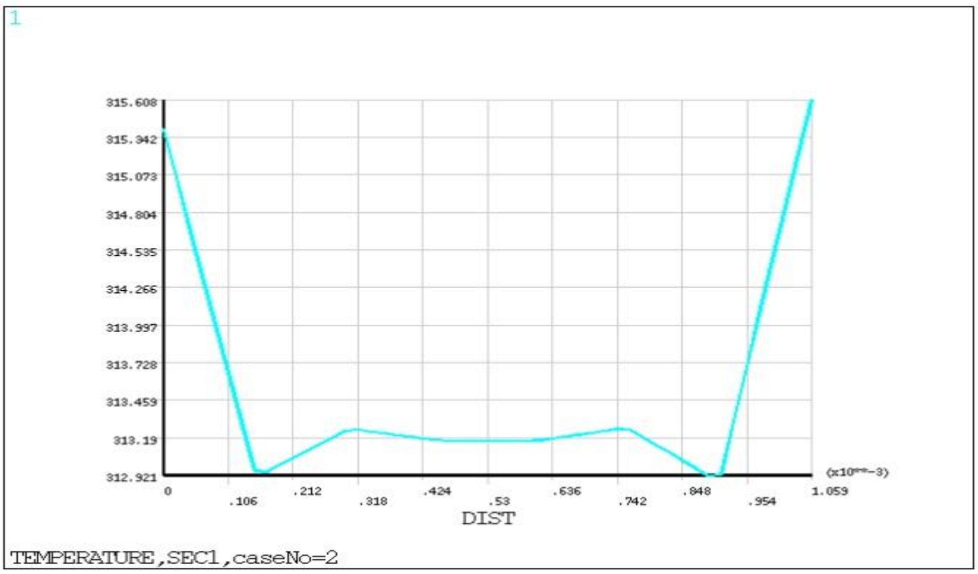

Figure 7. Graphical solution of fluid temperature at section (1)

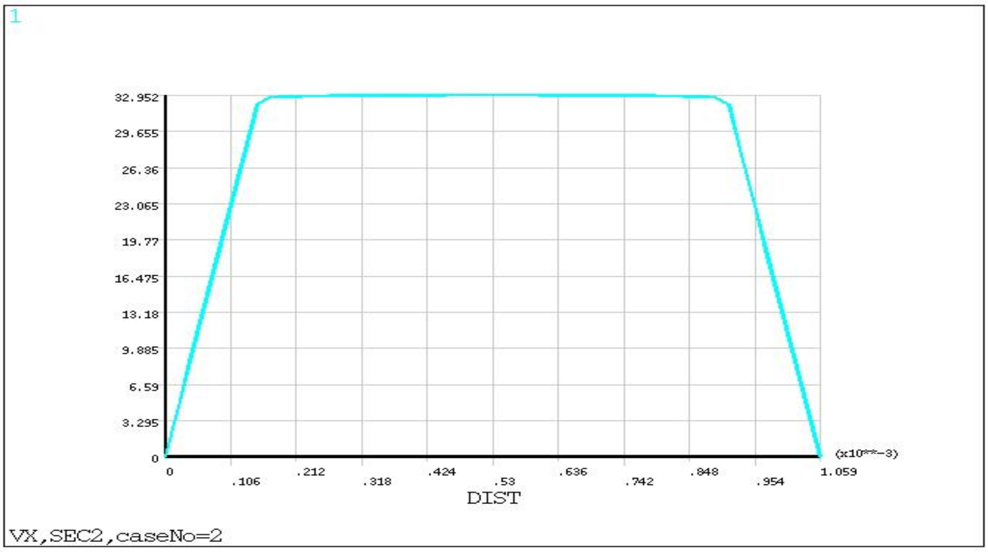

Figure 8. Graphical solution of fluid velocity at section (2)

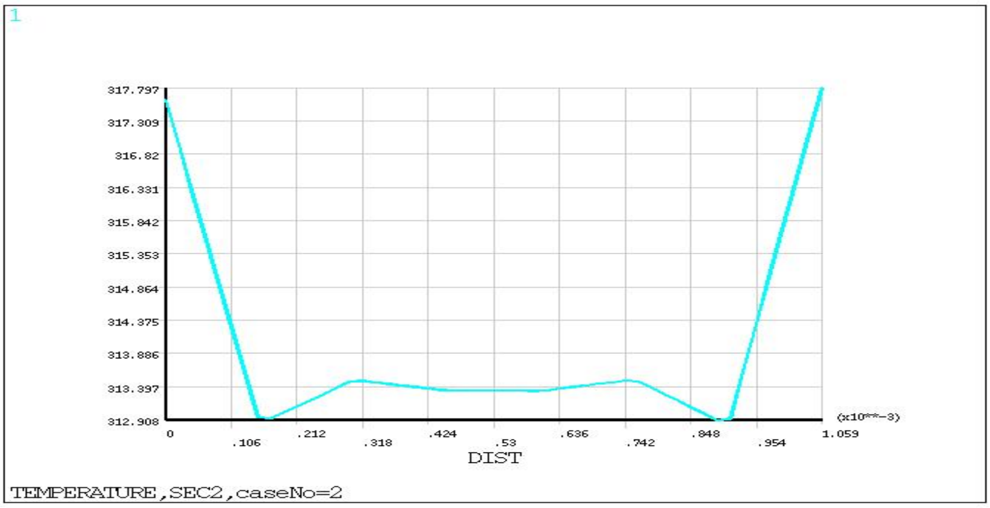

Figure 9. Graphical solution of fluid temperature at section (2)

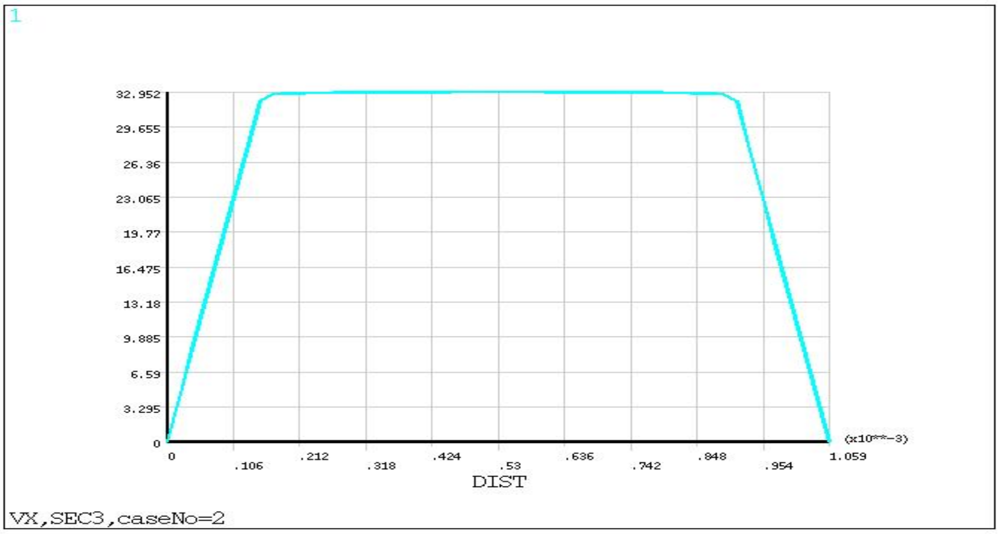

Figure 10. Graphical solution of fluid velocity at section (3)

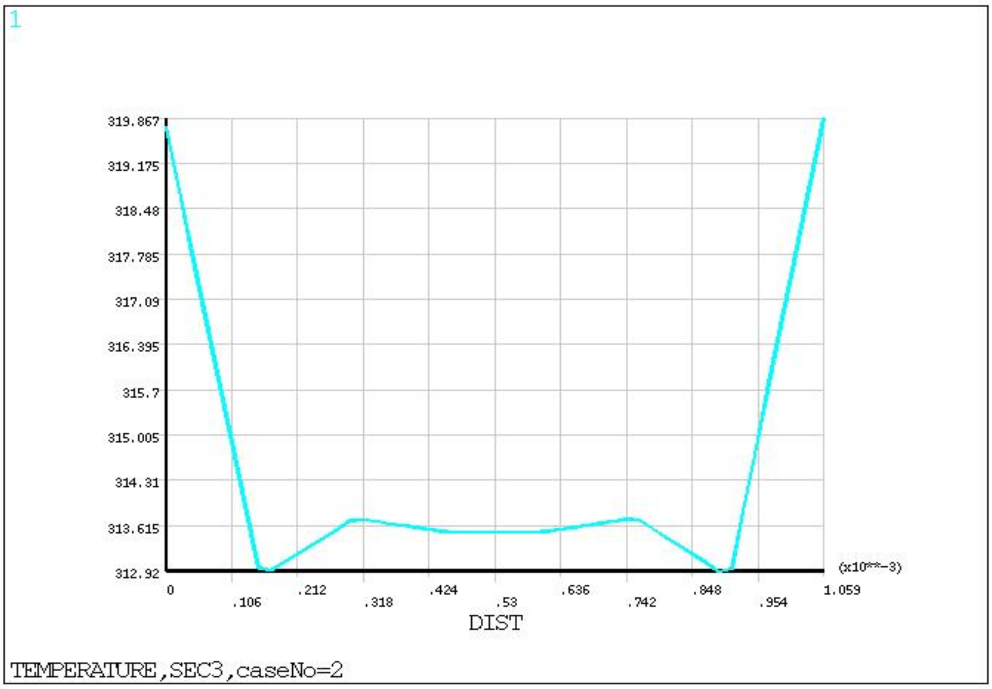

Figure 11. Graphical solution of fluid temperature at section (3)

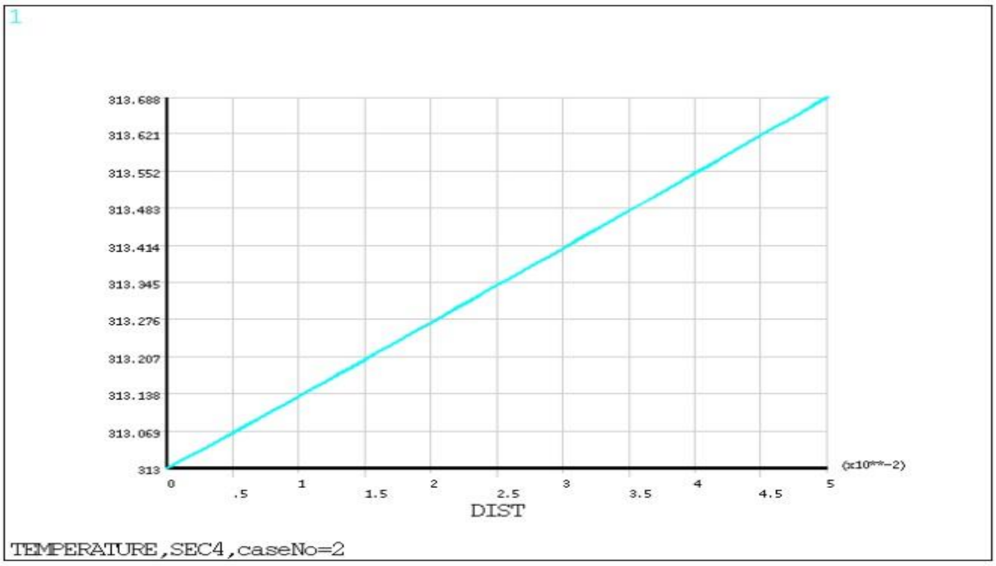

Figure 12. Graphical solution of fluid temperature at section (4)

4. Conclusions

1- Using ANSYS FLUENT software, the distributions of fluid velocity, static pressure and temperature can be obtained for a non-conventional machining system.

2- This simulation was done for a given experimental results where the experiments input data can be compared with the simulation results.

3- The distributions of fluid velocity, static pressure and temperature describe the fluid flow pattern into the interelectrode gap along flow path.

4- The distribution along the vertical solution (with respect to fluid flow direction) shows the turbulence of flow and increasing of fluid temperature along flow path.

5- For the parallel section (with respect to fluid flow direction), also confirm the increasing of fluid temperature toward the outlet flow.

5. References

1. ANSYS FlUENT(2009), "*ANSYS CFX-Solver Theory Guide*", ANSYS Inc.
2. Kozak J. (2012), "*Computer Simulation of Electrochemical Machining*", Proceedings of the World Congress on Engineering and Computer Science, Vol. 2,pp.1072-1084, San Francisco, USA.
3. Paczkowski T. (2011)," *Theoretical Analysis of Electromechanical Machining ECM Using Tool Electrode with Complex Translatory Motion*", Advances in Manufacturing Science and Technology Vol. 35, No. 1, pp. 72-85.
4. Mishra A., Jadhav D. B. and Jadhav P. V. (2014), "*CFD Analysis of Electrolyte Flow Pattern in Pulse ECM and to Optimize MRR for Circular Tool*" , International Journal of Engineering Research and General Science Vol. 2, No. 4. pp.17-24.
5. Zhiyong L. and Zongwei N. (2007),"*Convergence Analysis of the Numerical Solution for Cathode Design of Aero-engine Blades in Electrochemical Machining*" Chinese Journal of Aeronautics, Vol. 20, No. 6, pp. 570–576.
6. Christiansen S. and Rasmussen H. (1976), "*Numerical Solutions for Two-dimensional Annular Electrochemical Machining Problems*", Journal of Inst. Maths Applies, Vol.18, No.2, pp. 295-307.
7. De Silva A.K.M., Altena H.S.J., McGeough J.A. (2001), "*Influence of Electrolyte Concentration on Copying Accuracy of Precision- ECM*", International Journal of Engineering Research and General Science Vol.3. No.2, pp.28-37.
8. Ridha S. D. (2009), "*Theoretical and Experimental Study for Measurement of Residual Stresses Induced By Friction Stir Welding and Investigation of Stress Relieve By Vibration in Aluminum Alloy*", Ph.D. Thesis, College of Engineering, University of Baghdad, Iraq.