Journal of Engineering and sustainable Development
Vol. 21, No. 6, November 2017
ISSN 2520-0917
www.jeasd.org

NUMERICAL AND THEORETICAL STUDY ON THE EFFECT OF HEAT TRANSFER AND FLUID FLOW ON THE HYDRAULIC TANK

Dr. Rafa Abbas Hassan[1], Dr. Assel Khaleel Shyaa[2], *Ali Kadhim Khudadad[3]

1) Assist Prof., Mechanical Engineering Department, Al-Mustansiriyah University, Baghdad, Iraq.
2) Lecturer, Mechanical Engineering Department, Al-Mustansiriyah University, Baghdad, Iraq.
3) Mechanical Engineering Department, Al-Mustansiriyah University, Baghdad, Iraq

Abstract: Hydraulic systems will continue to play an important role in the industrial because of their large power to weight ratios , The paper introduce development of 128 liter hydraulic tank heat transfer and flow enhancement in order to reduce the heat that developed through the operation and keep fluid stability to reduce air bubble formation ,three cases are taken the first case with normal hydraulic tank with no addition the second with adding baffles and third case add diffuser and baffles the result show that there is enhancement in flow in the third case and enhancement in heat transfer in the second case , the theoretical result show good agreement in result of heat transfer in the first case.

Keywords: *Fluid flow hydraulic tank, Heat transfer*

دراسه عددية ونظرية على تأثير انتقال الحرارة وجريان المائع على الخزان الهيدروليكي

الخلاصة: الانظمه الهيدروليكيه سوف تستمر بلعب دور مهم في الصناعه بسبب القوه الكبيره نسبه الى الوزن , الدراسه تقدم تطوير خزان هيدروليكي بسعه 128 لتر من ناحيه تحسين انتقال الحراره وكذلك تحسين جريان المائع من اجل تقليل الحراره المتولده خلال تشغيل المنظومه وكذلك الحفاظ على استقراريه المائع من اجل تقليل تكون فقاعات الهواء , تم اخذ ثلاثه حالات الحاله الاولى خزان هيدروليكي بدون اي اضافه الحاله الثانيه باضافه قواطع والحاله الثالثه باضافه مشتت وقواطع النتائج تظهر هناك تحسين في الجريان في الحاله الثالثه وكذلك تحسين في انتقال الحراره في الحاله الثانيه , النتائج النظريه تظهر توافق جيد بنتائج انتقال الحراره بالنسبه للحاله الاولى.

1. Introduction

The using of fluid power mean using pressurized liquid to transmission power from location to another one , the hydraulic system has the main advantage ,force to torque can held constant , pressurized fluid can transmitted to long distance and overload protection .[1] the hydraulic tank play an important role in the hydraulic system when it's design in the proper way the main function of the hydraulic tank are ,cooling of the hydraulic fluid through the walls of the hydraulic tank by the surrounding area and the temperature difference by wall temperature and surrounding temperature and the designer of the hydraulic tank should increase the size of the area so the natural convection become better, remove air from the hydraulic system which one of the

* ali.kazem12@yahoo.com

problem in the hydraulic system and can cause the following effect -the formation of the bubbles which can noticed by the noisy sound operations in the components and it is occurs by excessive swirling of the hydraulic oil in the retune line of the hydraulic tank [2].

Loss of horsepower when the air bubbles is available in an actuator, it is alternately compressed and relaxed as the actuator is cycled. Since the air bubbles must first be compressed before the fluid can cause the actuator to move, power is consumed. Upon relaxation, the air pocket expands and rives fluid out. The stored power, therefore, is expended in driving fluid back into the reservoir and not in moving the actuator -Cavitations, defined simply as the formation of bubbles in a liquid, can have detrimental effects on a hydraulic pump. In an incorrectly designed hydraulic system, a vacuum may form on the hydraulic fluid, pulling trapped air out of the fluid to form small bubbles, [3],[4].

The summary of the previous work are finding the performance of six different types of fluids from four different manufacturers the mineral oil used is (iso VG 46) [5]. The previous work on the mathematical model for temperature distribution in hydraulic system has been developed [6] ,also efficient software modeling and simulation tolls offer an important series of option and views for simulation results [7].

2. Numerical simulation

The simulation process for both heat transfer and fluid flow within the hydraulic tank will discuss in this section the simulation which is done by using ANSYS FLUENT version 15 and have the following assumptions :

1-steady state condition

2-single phase fluid

3-constant properties at 40 C

4-steady flow rate

5-three dimensional

2.1. *the RNG turbulence model*

The k-epsilon models are the most widely validated turbulence models in literature and are the standard models to use in the commercial codes where the RNG model includes the following refinements:

- The RNG model has an additional term in its equation that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.

The RNG-based k-**ε** turbulence model is derived from the instantaneous Navies-Stokes equations. The analytical derivation results in a model with constants different from those in the standard k-**ε** model, and additional terms and functions in the transport equations for k-**ε**

The form of RNG k-**ε** are given below which are obtained from the following transport equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon \quad (1)$$

Dissipation rate of turbulence kinetic energy equation:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_i}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{K}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{K}-R_\varepsilon+s_3 \quad (2)$$

Where G_k is the generation of turbulence kinetic energy due to the mean velocity gradients; $G_b$is the generation of turbulence kinetic energy due to buoyancy;$C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$ are model constants; R_ε is a term which is related to the mean strain and turbulence quantities and represent the main difference between the RNG and standard k-ε Model this term is given by (Table 1) [9]

Table 1. Constant for the RNG (k-**ε**) model

C_μ	$C_{1\varepsilon}$	$C_{2\varepsilon}1$	σ_k	σ_ε
0.0845	1.42	1.68	1	1.3

Turbulence, in general, is affected by rotation or swirl in the mean flow. The RNG model in ANSYS Fluent provides an option to account for the effects of swirl or rotation by modifying the turbulent viscosity appropriately. The modification takes the following functional form

$$\mu_t=\mu_{t_0} f(\alpha_s, \Omega, \frac{k}{\varepsilon}) \quad (3)$$

Where μ_{t_0} is the value of turbulent viscosity calculated without the swirl modification using either from (1) or (2)

$\Omega$is a characteristic swirl number evaluated within ANSYS Fluent, and $\alpha_s$is a swirl constant that assumes different values depending on whether the flow is swirl dominated or only mildly swirling. This swirl modification always takes effect for ax symmetric, swirling flows and three-dimensional flows when the RNG model is selected. For mildly swirling flows (the default in ANSYS Fluent),α_s is set to 0.07. For strongly swirling flows, however, a higher value of $\alpha_s$can be used.[5]

2.2. *Geometry formation*

A three dimensional model of a hydraulic tank was represented by AutoCAD 2015 widely used CAD software. And were based on a real industrial 128litre tank built with dimensions of 800 × 400 × 400 mm, the main parts of the geometry shown in Fig. 1. As shown in Fig. 1. there is three cases with the following parts

(1) Return line (inlet port)
(2) Suction line (outlet port)
(3,4) Baffles (portion wall) with small opening at the lower

(5) Diffuser which have 18 holes with 5mm diameter to give sufficient flow.

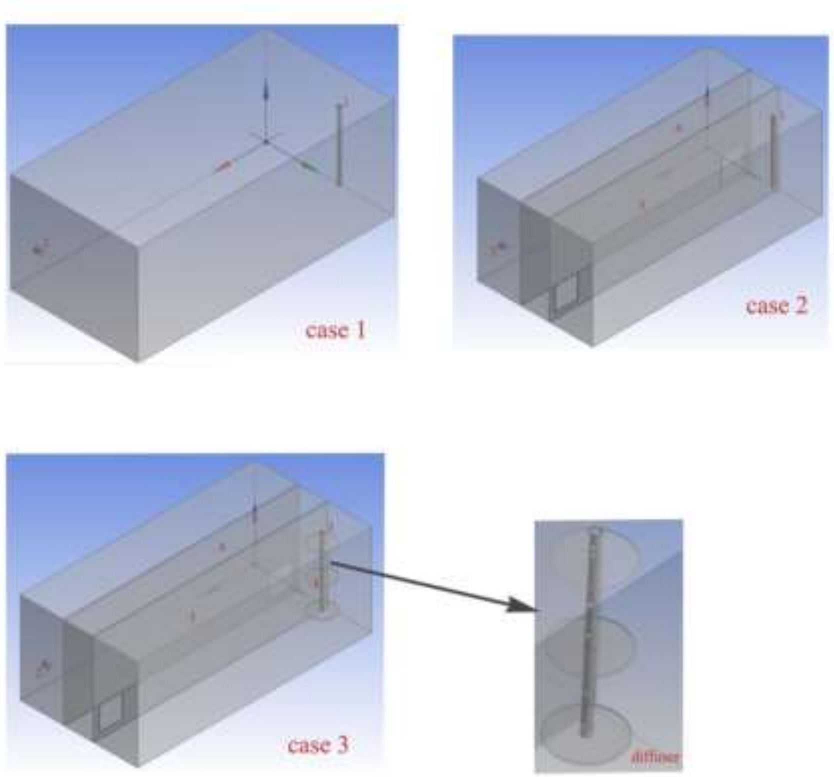

Figure 1. 3D model of the fluid inside the tank

Case 1: the return line is immersed vertically into the oil to reduce air bubbles and suction line is on the other side of the tank farthest point to the return line so the oil have sufficient distance to cool down.

Case 2: two longitudinal baffles was placed in the tank with small opening in the lower part since cold oil will be in the lower part the tank due to the density baffles are used so that the oil could take the longest bath in the tank noted that the 2^{nd} same to the first model with suction and return lines.

Case 3: technique of diffuser is used with the same baffles, the diffuser have (18) holes each one with (4 mm) in diameter .

2.3. *Mesh generation*

The area divide into the default setting selected by the fluent as show in Fig 2. Solvers the number of nodes and element according to each case with an element size of (8 mm) are show in (Table 2)

Table 2. the no. of element and nodes for each case

Cases	Case 1	Case 2	Case 3
Element size	8 mm	8 mm	8 mm
No. of Nodes	681846	688870	719118
No. of Element	3756535	3733843	3896188

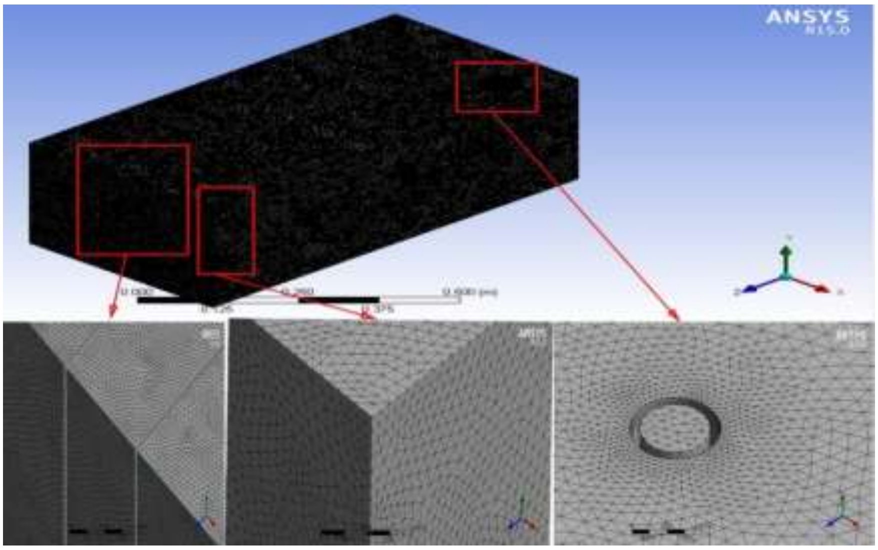

Figure 2. case 2 with element size 8 mm

The number of element that mentioned in Table (2) is the number that give the minimum error and also it is depend on the efficiency of the computer.

2.4. *Boundary condition and numerical solving*

Boundary conditions are specified for each zone of the computational domain as follow:

a .wall boundary condition:

The walls are divide according to the value of heat transfer coefficient calculated at the average value of the temperature the boundary condition is selected at mixed wall boundary condition with both convection and radiation and divide to (side wall, top wall and bottom wall) as show in Fig 3.

b. inlet and outlet boundary condition:

The momentum boundary condition is selected to be mass flow rate calculated to be 0.4 kg/sec and the outlet boundary condition is left to be calculated by the FLUENT at it is default setting.

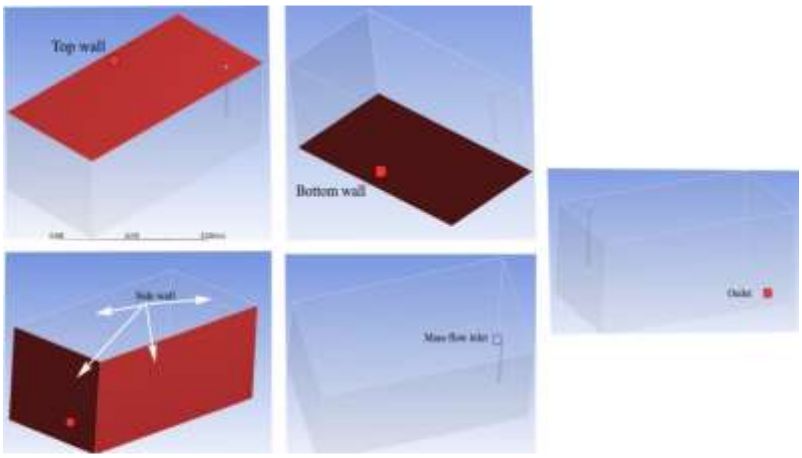

Figure 3. boundary condition

The convergence target was $10^{-6}$with the number of iteration was 500 and in the three cases the convergence achieved within 400 iteration and it is take 5-6 hours to achieve it with the residual show in Table (3).

Table 3. the residual value

	residual
Continuity	10^{-3}
X-velocity	10^{-3}
Y-velocity	10^{-3}
Z-velocity	10^{-3}
Energy	10^{-6}
K	10^{-3}
Epsilon	10^{-3}

3 .Theoretical calculation

128 liter tank is used with the following dimensions as show in Fig 4.
L=80cm ,B=40cm, H=40cm

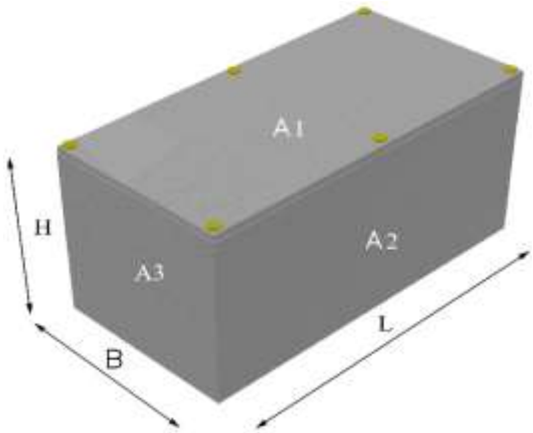

Figure 4. dimensions of the hydraulic tank

1- For the vertical wall

The heat transfer by conduction through vertical wall

$$Q_{vw}=\frac{2\,k}{s}\,(A_2 + A_3)(T_f - T_{vw}) \tag{3}$$

$$Q_{vw\;conv.} = 2\;h\;(A_2 + A_3)(T_{vw} - T_{\infty}) \tag{4}$$

Apply energy equation

$$Q_1=\frac{2\,k}{s}\,(A_2 + A_3)(T_f - T_{vw}) = 2\;h\;(A_2 + A_3)(T_{vw} - T_{\infty}) \tag{5}$$

Rearrange the equation

$$Q_1=\frac{(T_f-T_{vw})}{\frac{1}{\frac{2\,k}{s}(A_2+A_3)}}=\frac{(T_{vw}-T_\infty)}{\frac{1}{2\,h\,(A_2+A_3)}} \tag{6}$$

Eliminate T_{vw}

$$T_{vw} = Q_1 \times \frac{1}{2\,h\,(A_2+A_3)} + T_\infty \tag{7}$$

Sub equation (7) in equation(6)

$$Q_1=\frac{T_f - Q \times \frac{1}{2\,h\,(A_2+A_3)} - T_\infty}{\frac{s}{2k}(A_2+A_3)} \tag{8}$$

$$Q_1=\frac{T_f - T_\infty}{\frac{s}{2k(A_2+A_3)}+\frac{1}{2h(A_2+A_3)}} \tag{9}$$

1- For horizontal wall

By the same way of calculation

$$Q_2=\frac{T_f-T_\infty}{\frac{s}{kA_1}+\frac{1}{hA_1}} \tag{10}$$

1- For top surface

The heat transfers from the surface of the fluid in the tank to the atmosphere by natural convection are

$$Q_3 = h\,A_1\left(T_f - T_{\infty\,in}\right) \tag{11}$$

If the tank is open $T_{\infty\,in}$can assumed to be equal to the T_∞

The heat transferred by radiation can be determined by the Stefan-Boltzmann Law. It is found that the emissive power of a black body is directly proportional to the fourth power of its absolute temperature.

$$Q= \varepsilon\times\boldsymbol{\sigma}\times T^4 \tag{12}$$

Also it is noted that the energy emitted from one body, like the wall of a pipe section, is completely absorbed by the surroundings. Thus, the heat transferred from the body to its surroundings is given as

$$Q = \varepsilon\times\boldsymbol{\sigma}\times A\times (T_S^4 - T_\infty^4) \tag{13}$$

The effect of conduction, convection and radiation for both 1, 2, 3 Are

For vertical walls

$$Q_{11}=\frac{T_f-T_\infty}{\frac{s}{2k(A_2+A_3)}+\frac{1}{2h(A_2+A_3)}}+2\times\varepsilon\times\sigma\times(A_2+A_3)\,(T_{vw}^4-T_\infty^4) \tag{14}$$

For bottom of tank

$$Q_{22}=\frac{T_f-T_\infty}{\frac{s}{kA_1}+\frac{1}{hA_1}}+\varepsilon\,\sigma\,A_1(T_{vw}^4-T_\infty^4) \tag{15}$$

For top surface

$$Q_{33} = h\,A_1\left(T_f - T_{\infty\,in}\right) + \varepsilon\,б\,A_1\,(T_{wt}^4 - T_{\infty}^4) \tag{16}$$

Where:
T_{wt} Top wall temperature
The total heat transfer from the tank walls are in the following equation [8]

$$Q_t = Q_{11}+Q_{22}+Q_{33}\ \text{(W)} \tag{17}$$

4 . Result and discussion

The simulation result show in three dimensional stream line are show in Fig 4. for case 1 that give indication for the fluid flow inside the hydraulic tank , from the figure that the fluid in the return line is impingement in the floor of the hydraulic tank and increase the turbulence and that cause increasing air bubbles and in Fig 5. the hot fluid is restricted near the return line

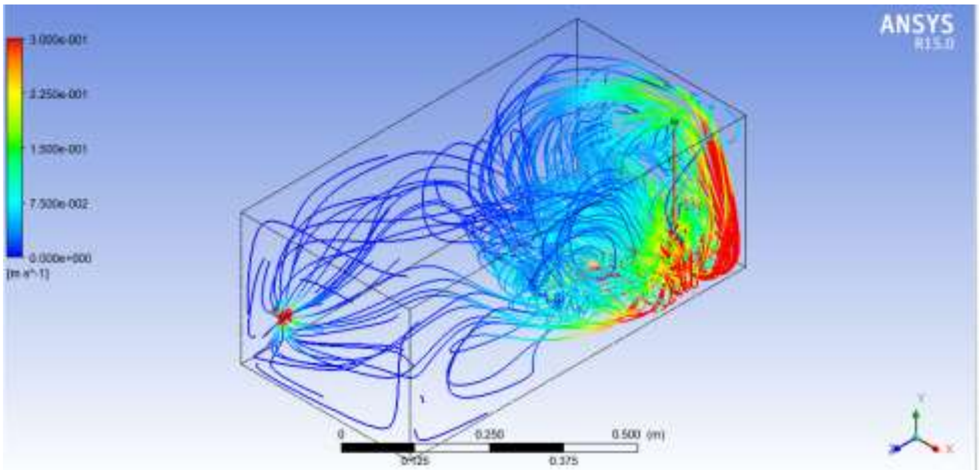

Figure 4. case 1 streamlines at No. of points 100

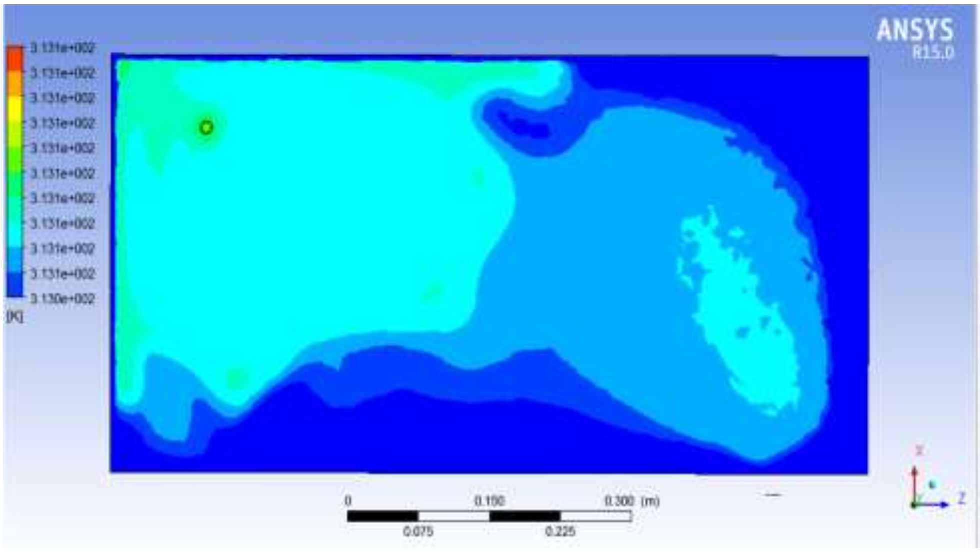

Figure 5. temperature distribution x-z plane top view at x=330 mm from the bottom case1

While in case 2 the fluid flow is impingement in the floor , baffle and side wall of the hydraulic tank and the long bath of the hydraulic oil from the return line to the suction line let the air baubles release through the long path since the inertia force of the air is small if compared with the inertia force of the hydraulic oil and that will allow the air bubbles goes up while moving ,also the heat is distributed through the hydraulic tank if compared to case 1 and more heat is dissipated through the hydraulic tank since the hydraulic oil will take the longest path to reach to the suction line as show in Fig. 6.

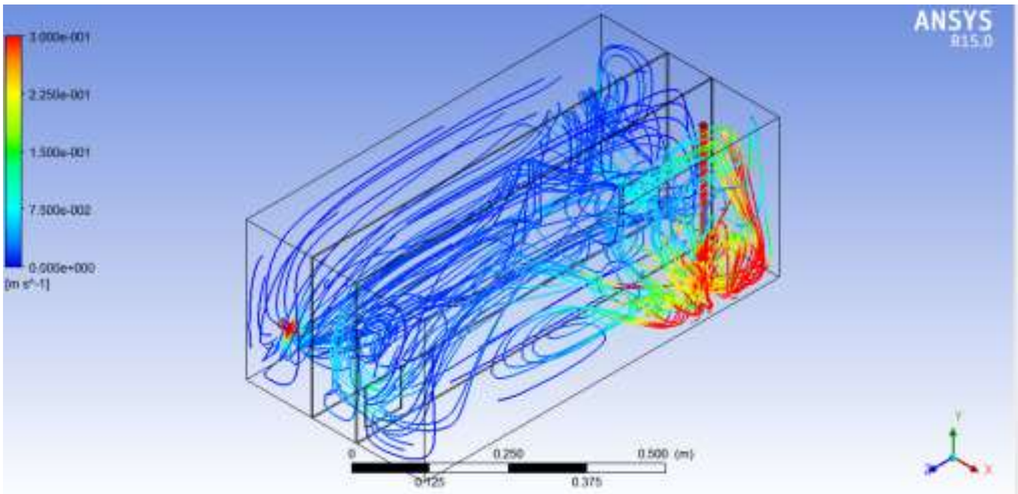

Figure 6. case 2 streamlines at no. of points 100

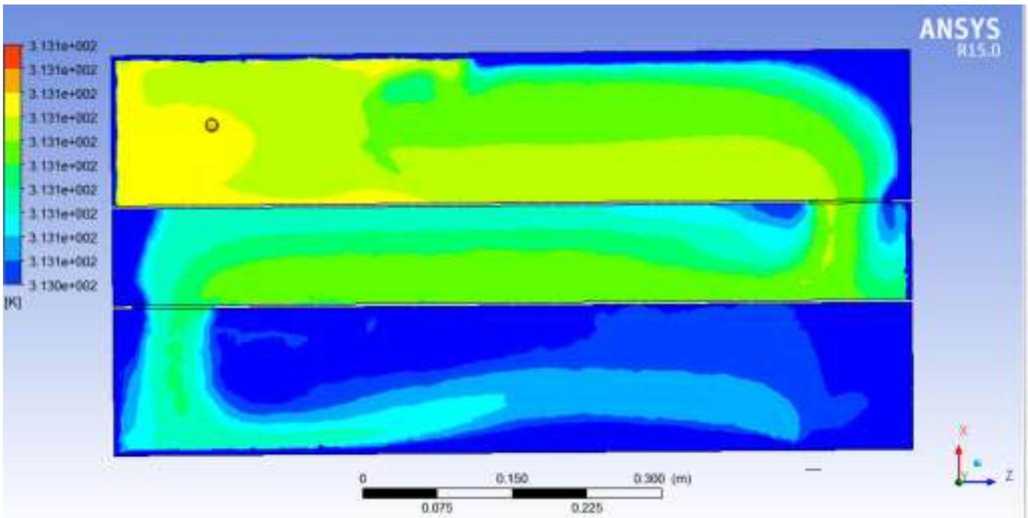

Figure 7. temperature distribution x-z plane top view at x=330 mm from the bottom case 2

In case 3 the fluid flow is distributed through the diffuser with stable stream line as show in Fig 8. that will led for more air baubles to release through the hydraulic oil since air bubbles can't be available at the hydraulic oil streamlines according to Table 4. less heat transfer if compared to case 2 as show in Fig 9.

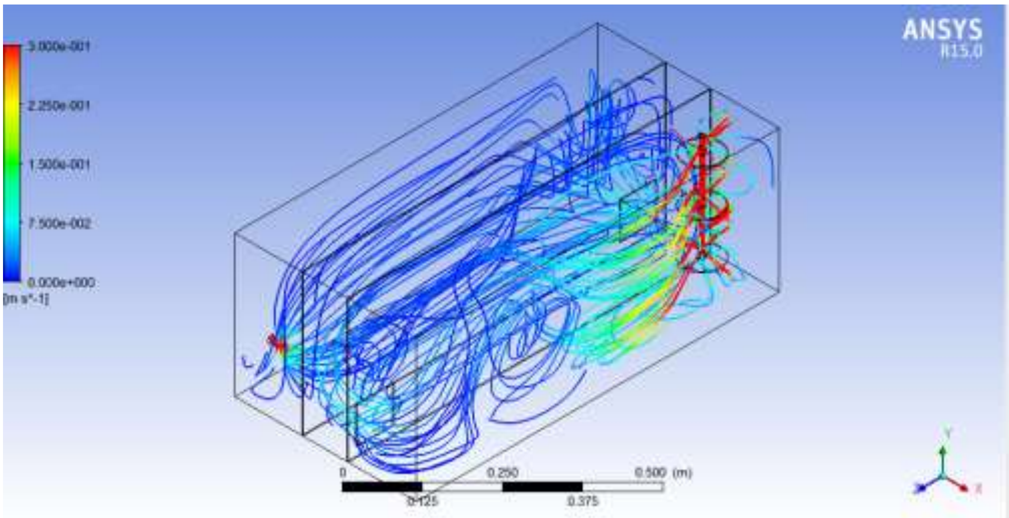

Figure 8. case 3 streamlines no. of points 100

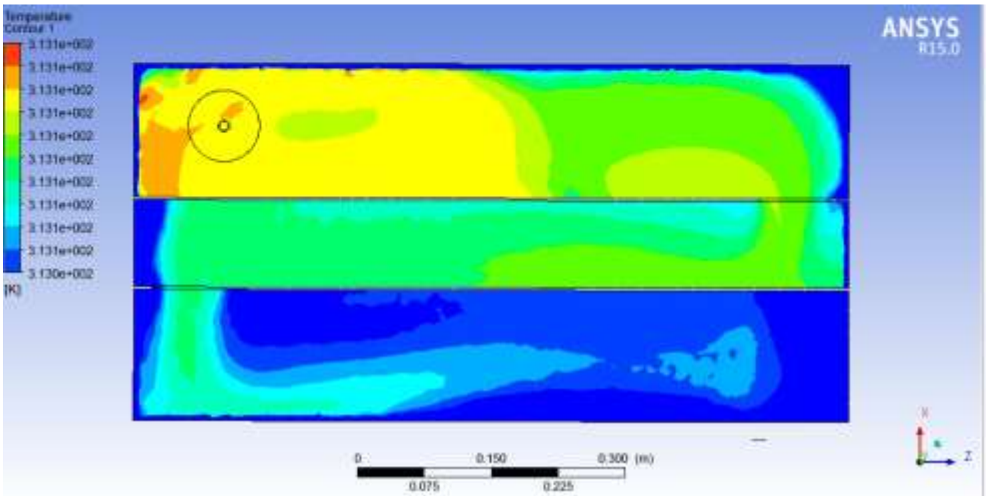

Figure 9. Temperature distribution x-z plane top view at x=330 mm from the bottom case 3

5. Conclusions

1-There is good agreement between the result of heat transfer between the numerical and theoretical solution and numerical simulation in heat transfer according to case 1 (standard hydraulic tank).

2-The simulation result show that case 2(baffle tank) is slight better than case 3in heat transfer.

3- The simulation result show that case 3(baffle and diffuser) is better than case 2 in fluid flow stability.

Abbreviations

S wall thickness (m)
k Thermal conductivity (w/m. k)
T_f Fluid temperature

T_{vw} Vertical wall temperature
And the outside wall tank that surrounding atmosphere by natural convection
h Heat transfer coefficient (w/$m^2.k$)
T_∞ Atmospheric temperature
Q_1 Heat transfer from fluid through the vertical wall by conduction and convection.
The Stefan-Boltzmann constant has the value of 5.6 7x10^{-8}(W/m.k^4)σ
ε The emissivity of the body (dimensionless less)
Q_2 Heat transfer from fluid through the bottom of the tank by conduction and convection.
$T_{\infty\ in}$ The temperature of air inside the tank

6. References

1. Akers, A., Gassman, M. and Smith, R. (2006), "*hydraulic power system analysis*", Taylor and Francis group, New York.
2. Daddannaavar, R., and Branard, A. (2005), "*practical hydraulic system*"Elsevier, Netherland.
3. Totten, G. E. and De Negri, V. J., (2012), "*Handbook of Hydraulic Fluid Technology*", Taylor and Francis group, Second Edition, New York.
4. Hehn, A. H., (1993) " *fluid power troubleshooting* " second edition,Gulf publishing company, New York
5. Santosh Javalagi (2012) "*Hydraulic fluid properties and its influence on system performance*."Division of fluid and mechatronic systemsDepartment of management and engineering
6. Al-Natour. (1992) "*study of an open circuit hydraulic power system with compact cooler reservoir unit*" Dublin City University
7. Vito Tič and Darko. (2012) " *design of modern hydraulic tank using fluid flow simulation* " university of Maribor, faculty of mechanical engineering si- 2000 Maribor ,Slovenia
8. YUNUS A. CENGEL. (2014) "*heat transfer*" particle approach 2nd edition ISBN-13: 978-0072458930
9. ANSYS FLUENT 15.0, Theory Guide, (2014).