Modeling and Calculation of Coupled Heat Transfer for Heating Process of Perforating Test Tool

Yongqiang Hou*, Guangzheng Jiaa

School of Mechanical Science and Engineering, Northeast Petroleum University, Daqing, China

*Corresponding author e-mail: train_1@163.com, jiaguangzheng@126.com

Abstract. The perforating bullet detection experimental device is used to simulate the downhole high-temperature and high-pressure environment in which the perforating bullet performance is test. The perforating test tool is inside liquid-filled cylindrical wellbore cavity that is heated by heating wellbore wall with natural convection heat transfer. A mathematical model for unsteady heat-fluid-solid coupling heat transfer between the liquid in the wellbore cavity and the perforating test tool was established. The projection method is used to numerically solve the mathematical model, and the results of the dynamic heating evolution process of the perforation experimental device are calculated. The internal flow field and temperature field of the perforation experimental device with and without the perforating test tool in the wellbore are analyzed. Through the calculation and analysis results, it is found that the perforating test tool inside the simulated wellbore cavity has a greater influence on the flow field and temperature field.

1. Introduction

The oil and gas well perforation is very important in oil and gas exploration and production period. With the well depth increasing, the temperature and pressure of downhole reservoir increase correspondingly, which puts forward higher requirements for perforating projectile resisting higher temperature and pressure. The perforating bullet detection experimental device used for simulating the high-temperature and high-pressure downhole environment is investigated. The structure of experimental device is shown in figure 1. The device can provide some downhole simulation test conditions for the research and design of the perforating bullet [1]. The perforating test tool is installed inside the liquid-filled cylindrical simulated wellbore enclosed cavity. The vertical wellbore wall is heated by the pit furnace, and the internal fluid and the perforating test tool are heated by heat conduction. The heat exchange process involves simulating the natural convection of fluid in the wellbore and the coupled heat transfer between perforating test tool and fluid. Most domestic and foreign scholars' research on the above problems are simplified to the natural convection coupled heat transfer process in a closed square cavity with given internal boundary conditions. The square cavity structures are mainly conducted because of modeling and numerical calculation easily. According to the solution results of the flow field and temperature field in the simulated wellbore, cylindrical cavity structure is more accurate. It can optimize heating process of the perforating test tool and improve heating control accuracy, which has certain theoretical and engineering application value [2-5].
2. Physical model

As shown in Figure 1, the simulated wellbore heating system is mainly composed of a well-type heating furnace, simulated wellbore, wind circulation system and supporting base. The simulated wellbore is a high-strength cavity thick-walled metal cylinder that can withstand the pressure impact generated by the perforating test tool exploding inside. The pit-type electric heating furnace is a cylindrical furnace body with a vertical circular hearth. The air circulation device includes a fan, a circulation pipe, an internal circulation valve, an air inlet valve and a cooling valve. The pit-type electric heating furnace realizes uniform heating of the outer wall of the simulated wellbore by adjusting the heating power of the electric heater in the furnace.

Figure 1. Structure of heating device

The perforating test tool is installed inside the simulated wellbore cavity filled with liquid, and the high temperature simulated wellbore wall heats the internal fluid. Due to the uneven temperature of each part, the internal fluid forms a density difference, which generates buoyancy in the gravity field to form natural convection. The fluid and the perforating test tool perform heat fluid-solid coupling heat transfer. The simulated wellbore is a cylinder, the heating temperature field is radial symmetry, and the heat transfer process can be described by a two-dimensional model. Figure 2 shows the physical model of coupled heat transfer for the two-dimensional simulation of fluid and perforating test tool in the wellbore. $r$ is the coordinate axis in the horizontal radius direction, and $z$ is the coordinate axis in the vertical height direction (symmetry axis). The horizontal wall surface is an insulation surface, and the vertical wall surface is a high temperature surface.

Figure 2. Physical model of perforating experimental device
3. Coupling heat transfer mathematical model

3.1. Mathematical model
The two-dimensional unsteady thermal fluid-solid coupling heat transfer mathematical model between
the liquid and the perforating test tool in the cylindrical coordinate system uses the following two
assumptions: (1) the fluid is incompressible, (2) all physical parameters of water treat as a constant.
The energy equation is solved using the whole-field discrete and whole-field solution methods. The
respective density of the fluid and solid regions is multiplied by their specific heat [6]. The
dimensionless density in the fluid region, Fluid region is

\[ \frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial r} + \frac{\partial \rho v}{\partial z} = 0 \]  

momentum equation

\[ \frac{\partial (\rho v)}{\partial r} + \frac{\partial \rho v}{\partial z} + \frac{\partial \rho v}{\partial \theta} = - \frac{\partial p}{\partial r} + Pr \left( \frac{\partial^2 \rho v}{\partial r^2} + \frac{1}{r} \frac{\partial \rho v}{\partial r} + \frac{\partial^2 \rho v}{\partial z^2} - \frac{v}{r^2} \right) \]  

energy equation

\[ \frac{\partial (\Theta)}{\partial \theta} + \frac{\partial \Theta}{\partial r} + \frac{\partial \Theta}{\partial z} = M \left[ \frac{\partial^2 \Theta}{\partial r^2} + \frac{1}{r} \frac{\partial \Theta}{\partial r} + \frac{\partial^2 \Theta}{\partial z^2} \right] \]  

\( \tilde{z} \) and \( \tilde{r} \) represent dimensionless coordinates in vertical and horizontal directions respectively. Select
width \( H \) as characteristic length. \( T_{h-T_c} \) is the characteristic temperature. \( r = r/H \) and \( z = z/H \) are
dimensionless lengths. \( t = t(H/u_\cdot) \) is dimensionless time. \( u = u/u_\cdot \) and \( v = v/v_\cdot \) correspond to the
dimensionless velocities in the \( y \) and \( r \) directions (where \( u_\cdot = a/H \)). \( \Theta = (T - T_c)/(T_{h-T_c}) \) is dimensionless
temperature. \( \tilde{p} = p/\rho u^2 \) is dimensionless pressure. \( v \) is kinematic viscosity, \( m^2/s \). \( a \) is thermal
diffusivity, \( m^2/s \). \( Pr = \nu/\alpha \) is Prandtl number. \( g \) is acceleration of gravity. \( m^2/s \). \( \beta \) is thermal coefficient
of expansion, \( K^{-1} \). \( Ra = g \beta \lambda/(H - T_c) \) is Rayleigh number. \( a = \lambda/c_p \), \( \lambda \) is Thermal Conductivity,
\( W/(m-K) \). \( c_p \) is specific heat \( J/(kg-K) \). \( \rho \) is density, \( kg/m^3 \). \( a_f \) is thermal diffusivity in the fluid region,
\( m^2/s \). \( a_s \) is regional thermal diffusivity of solids, \( m^2/s \). The thermal diffusivity ratio of the solid to the
fluid region is \( a_f/a_s = n_f \). The coefficient of a dimensionless energy equation is \( M = a_f/(H \times u_\cdot) \). In the
dimensionless fluid region \( M = a_f/(H \times u_\cdot) = 1 \), In the solid region \( M = a_f/(H \times u_\cdot) = a_s \times n_f/(H \times u_\cdot) = n_s \).
3.2. Boundary conditions

Upper horizontal heat insulation surface, \( \dot{z}=1, \dot{u} = \dot{v} = 0, \partial \Theta / \partial z = 0 \). Lower horizontal heat insulation surface, \( \dot{z}=0, \dot{u} = \dot{v} = 0, \partial \Theta / \partial z = 0 \). High temperature vertical wall, \( r=1, \dot{u} = \dot{v} = 0, \Theta = 1 \). Symmetry axis, \( \dot{r}=0, \partial \dot{v} / \partial r = 0, \partial \Theta / \partial r = 0, u = 0. \) The velocity in the solid region, \( u=v=0 \).

4. Equation solving method

4.1. Discrete solving method

The calculation area inside the simulated wellbore is discretized by a staggered grid, divided into \((N-1)\) and \((M-1)\) equal parts, and \(N\) and \(M\) spatial nodes are obtained. \( n \) represents the non-steady state time horizon. Discrete equations (1) to (4) using the finite difference method, the convection terms are explicitly processed using the second-order Accamas-Bashforth format, the viscous terms are discretized using the second-order central difference format.

\[ \frac{\bar{u}_{i,j}^n - \bar{u}_{i,j}^{n-1}}{\Delta t} + \frac{\bar{v}_{i,j}^n - \bar{v}_{i-1,j}^n}{\Delta r} + \frac{\bar{v}_{i,j}^n}{\bar{r}} = 0 \]  

\[ \frac{\bar{u}_{i+1/2,j}^{n+1} - \bar{u}_{i,j}^n}{\Delta t} + D_z (\bar{u}\bar{v})_{i,j}^n + D_t (\bar{v}\bar{u})_{i,j}^n = -\left( \bar{p}_{i+1,j} - \bar{p}_{i,j} \right) + p_i \left[ L_n (\bar{v})_{i,j}^n + \frac{1}{\bar{r}} D_t (\bar{v})_{i,j}^n - \left( \frac{\bar{v}}{\bar{r}} \right)_{i,j}^n \right] \]  

\[ \frac{\bar{u}_{i,j}^{n+1} - \bar{u}_{i,j}^n}{\Delta t} + D_z (\bar{u}\bar{v})_{i,j}^n + D_t (\bar{v}\bar{u})_{i,j}^n = -\left( \bar{p}_{i+1,j} - \bar{p}_{i,j} \right) + p_i \left[ L_n (\bar{u})_{i,j}^n + \frac{1}{\bar{r}} D_t (\bar{u})_{i,j}^n - \frac{1}{\bar{r}} D_r (\bar{u})_{i,j}^n \right] - PrRa(\bar{\Theta})_{i,j} \]  

\[ \frac{\bar{\Theta}_{i,j}^{n+1} - \bar{\Theta}_{i,j}^n}{\Delta t} + D_z^2 (\bar{u}\bar{\Theta})_{i,j} + D_t^2 (\bar{v}\bar{\Theta})_{i,j} = L_n (\bar{\Theta})_{i,j} + \frac{1}{\bar{r}_{i,j}} D_r (\bar{\Theta})_{i,j} \]  

The difference operator is defined as [7]:\( D(\ast)_{i,j} = ((\ast)_{i+1/2,j} - (\ast)_{i-1/2,j}) \Delta r \), \( D(\ast)_{i,j} = ((\ast)_{i,j+1} - (\ast)_{i,j-1}) \Delta z \), \( D(\ast)_{i,j} = ((\ast)_{i+1,j} + (\ast)_{i-1,j}) / 2 \), \( D(\ast)_{i,j} = ((\ast)_{i,j+1} + (\ast)_{i,j-1}) / 2 \), \( L_n(\ast)_{i,j} = ((\ast)_{i+1,j} + (\ast)_{i,j+1} + (\ast)_{i-1,j} - 2(\ast)_{i,j}) / 4 \), \( \Delta r \) and \( \Delta z \) are the space steps in \( r \) and \( z \) directions, respectively.

4.2. Projection method

The projection method is used to solve the mathematical model. The projection method can very efficiently solve the Navier-Stokes equations of unsteady incompressible flow. The projection method was originally proposed by Chorin [8], which is based on the Helmholtz-Hedge vector decomposition theorem [9]. The projection method divides the decoupled Navier-Stokes equation into four steps for solving equations, and obtains the velocity and pressure values. Using Helmholtz-Hedge vector...
decomposition method, by introducing an intermediate velocity $V^*$, momentum equation is divided into the sum of two operators:

$$\frac{V^* - V^n}{\Delta t} + (V \cdot \nabla V)^n = Pr \Delta^2 V^n - Pr Ra j \Theta^n$$  \hspace{1cm} (9)$$

$$\frac{V^{n+1} - V^*}{\Delta t} = -\nabla P^{n+1}$$  \hspace{1cm} (10)$$

The first step, the prediction step, solves the intermediate velocity field $V^*$ by formula (9).

The second step, the pressure correction step, discrete momentum equation substituted into the discrete continuity equation to obtain the discrete pressure Poisson equation.

Solve the pressure $P^{n+1}_{i,j}$ value using successive over relaxation (SOR) method, $m$ is the number of iterations, $\omega > 1$ is the relaxation factor. The difference lattice of super-relaxation iteration is

$$p_{i,j}^{n+1} = \omega \frac{b p_{i+1,j}^{m+1} + c p_{i-1,j}^{m+1} + d p_{i,j+1}^{m+1} + e p_{i,j-1}^{m+1} + f}{a} + (1 - \omega) p_{i,j}^{n}$$  \hspace{1cm} (11)$$

$$a = \frac{\Delta t}{\Delta x^2} + \frac{\Delta t}{\Delta x^2} + \frac{\Delta t}{\Delta r^2} + \frac{\Delta t}{\Delta r^2} + \frac{\Delta t}{\Delta r^2}, \quad b = -\frac{\Delta t}{\Delta r^2} - \frac{\Delta t}{\Delta r^2}, \quad c = -\frac{\Delta t}{\Delta r^2}, \quad d = -\frac{\Delta t}{\Delta x^2}, \quad e = -\frac{\Delta t}{\Delta x^2},$$

$$f = \frac{u_{i,j}^{n+1} - u_{i,j}^{n}}{\Delta x} + \frac{v_{i,j}^{n+1} - v_{i,j}^{n}}{\Delta r} + \frac{v_{i,j}^{n+1} - v_{i,j}^{n}}{r}.$$  \hspace{1cm} (12)$$

The third step is to obtain the value of $V^{n+1}$ at the time $n+1$ by applying the formula (10) from the obtained $V^*$ and $P^{n+1}$ values.

The fourth step is apply formula (8) from the obtained $V^{n+1}$ value to obtain the $\Theta^{n+1}$ value at $n+1$ time.

5. Model verification

In order to verify the correctness of the established mathematical model, the numerical calculation uses the physical parameters same as parameters of the experimental device. Experimental and simulation calculation of the heating curve is shown in Figure 3. The simulation calculation results are basically consistent with the experimental results, both of which are slow in the initial heating period, then the temperature rises rapidly, and the top temperature is slightly higher than the bottom temperature. In the initial heating period of 0~1.5h, the maximum relative error at the top is 19%, and the maximum relative error at the lower part is 16%. With the increase of heating time, the relative error decreases, and the relative error at the top and the bottom is within 6%.

![Figure 3](image_url)

**Figure 3** The measured and simulated calculation of the heating curve
6. Calculation results and analysis

6.1. Calculation parameters
According to the established mathematical model and numerical solution method, a calculation program is written to perform numerical calculation on the heating process of the fluid in the simulated wellbore and the perforating test tool. Simulated wellbore cavity radius \( r = 0.175 \) m, height \( H = 2.5 \) m. The high-temperature wall surface is 400°C. The fluid in the simulated wellbore is water, \( Pr = 0.93 \), \( Ra = 5.87 \times 10^{9} \). According to the grid independence test, the number of nodes in the radial direction is \( N = 33 \), and the number of nodes in the height direction is \( Z = 449 \). The CFL condition in the radius and height directions are less than 0.1, which satisfies the stable conditions for flow field calculation. The dimensionless fluid area \( M = 1 \), the perforating test tool area \( M = 7.10 \).

6.2. Calculation result analysis
For intuitive expression, according to the dimensional conversion relationship of the mathematical model, the calculation results are expressed in dimensional form. The high-temperature wall of the cylindrical cavity heats the fluid in the closed cavity to form natural convection heat transfer. Figure 4 (a) is a fluid velocity field of a simulated wellbore cavity without a perforating test tool. The fluid inside the simulated wellbore forms an annular natural convection motion. Figure 4 (b) shows that there is a perforating test tool inside the simulated wellbore cavity. Due to the function of the perforating test tool, the area where the gap between the test tool and the inner wall is large forms a natural convection motion that forms three small rings.

Figure 5 (a) is the temperature field distribution without the perforating test tool (Heating time is 6h). It can be seen that the temperature of the internal fluid in the simulated wellbore is lower than that of the bottom under the effect of natural convection. Figure 5 (b) is the temperature field distribution with a perforating test tool (Heating time is 6h). It can be seen that due to the heat transfer coupling effect of the internal fluid and the test tool, the gap between the test tool and the inner wall of the simulated wellbore is large, and the heat is directed to the axis slow conduction. Where the gap between the test tool and the inner wall of the simulated wellbore is small, the heat is transferred faster to the axis.

![Figure 4](image_url)

**Figure 4.** (a) fluid velocity field without test tool. (b) fluid velocity field with test tool.
Figure 5. (a) temperature field distribution without perforating test tool (Heating time is 6h). (b) temperature field distribution with perforating test tool (Heating time is 6h).

Figures 6 (a) and 6 (b) show the radial temperature distribution at the position where h=1.78m, respectively, without or with perforating test tool inside the simulated wellbore.

Figure 6. (a) temperature distribution at h=1.78m, without perforating test tool (b) temperature distribution at h=1.78m, with perforating test tool

It takes about 10 hours to heat the central position of without perforating test tool to 200°C, and about 4.9 hours when the central position of with test tool is heated to 200°C. The temperature rise rate when the test tool inside is faster than without it.

7. Conclusion
The dynamic coupling heat transfer process of the fluid in the simulated wellbore and the perforating test tool in the cavity was studied. Concluded as follow:

(1).Based on the analysis of the coupled heat transfer process in the simulated wellbore, a coupled heat transfer mathematical model of the simulated wellbore and its internal perforating test tool was established. The finite difference method is used to discretely simulate the wellbore coupled heat transfer mathematical model, and the projection method is used to numerically solve the established mathematical model.

(2) The comparison between the experimental data and the simulation calculation results verifies the correctness of the established mathematical model to solve the heat transfer process of the perforating test tool.
(3) The installation of the perforating test tool inside the simulated wellbore cavity has a greater influence on the flow and temperature fields. The temperature rise rate when the perforating test tool is inside is faster than when there is without it.

Acknowledgments
This work was financially supported by Scientific research operating expenses of provincial universities in Heilongjiang province fund.

References
[1] Y G Liu. Research on ultra-high temperature deep penetration transmission hole device. Qingdao: China University of Petroleum (East China), 2013.
[2] M Y Ha, M J Jung, Y S Kim. A numerical study on transient heat transfer and fluid flow of natural convection in an enclosure with a heat-generating conducting body. Numerical Heat Transfer, Part A, 1999, 35: 415-434.
[3] R B Yedder, E Bilgen. Laminar natural convection in inclined enclosures bounded by a solid wall. Heat and Mass Transfer, 1997, 32: 455-462.
[4] B Calgagni, F Marsili, M Paroncini. Natural convective heat transfer in square enclosures heated from below. Applied Thermal Engineering 2005, 25: 2522-2531.
[5] O Aydin, W J Yang. Natural convection in enclosures with localized heating from below and symmetrical cooling from sides. Int. J. Numer. Methods Heat Fluid Flow, 2000, 10(5): 519-529.
[6] P X Jiang, D Y Ke, Z X Ren. Study on the Combination of Unsteady Heat Conduction with Natural Convection and Natural Convection and Radiation Heat Transfer with Internal Heat Source. Chinese Journal of Computational Physics, 1999, 16 (3): 302-308.
[7] M E Liu, Y X Ren, H X Zhang. Approximate projection methods for incompressible flow on nonstaggered grids. Acta Aerodynamica Sinica, 2005, 23(3): 273-278.
[8] J. Guermond, P. Minev, J. Shen. An overview of projection methods for incompressible flows. Computer Methods in Applied Mechanics and Engineering 2006, 195: 6011–6045.
[9] J H Ferziger, M Peric. Computational methods for fluid dynamics. Berlin, Springer 1996.