Numerical and experimental investigation of heat transfer in cable heated pipeline

R Deeb and D V Sidenkov
Department of theoretical basis of heat engineering, National Research University “Moscow Power Engineering Institute”, Russia, 111250 Moscow, Krasnokazarmennaya, 14
E-mail: e.rawad.deeb@yandex.com; sidenkovdv@mpei.ru

Abstract. Nowadays due to the development of computer technology and the creation of specialized software packages 2-D and 3-D the design of technical devices and modeling their operations in various operating systems is practically available for technical engineers. This work considers the possibility of studying the numerical simulation technology in the ANSYS package for the temperature distribution of a thermally-insulated pipeline equipped with a cable heating system.

1. Introduction
Pipelines which carry the heated fluids is the main type of heat and mass transfer in various heat exchangers and technological systems to support human activities. In Russian climate conditions, it is very important to know how to maintain the thermal domain of such systems. In the educational process of MPEI, students of thermal and power specialization have performed thermal and hydraulic calculations of various pipe (channel) systems such as standard calculations and reports in the preparation of bachelor's and master's dissertations.

One of the ways to maintain the thermal domain is to use electrical cable heating technology. Cable heating systems protect tanks and reservoirs, aboveground and underground pipelines from low temperatures, pipes and oil pipelines which are used to pump fluids and chemicals from freezing.

The actual problems are the optimization of energy consumption when using cable systems to maintain a stable temperature of the transported substance (heat loss compensation), warming up the “stopped” pipeline, heating the moving environment up to the set temperature, determining the optimum length of the warmed area.

The use of an electrical cable heating pipelines can have a number of technological advantages, namely, the capability of installation in the location, the compact design, equipment connection, acceptable capital costs and automation control process.

To ensure efficient and safe operation, electric heating systems are equipped with monitoring and control systems. For example, based on the temperature sensors of the transferred fluids, heating element and the environment, the signal and protection systems interfere at the right time.

Nowadays, due to the increase of computers speed, it has become possible to create predictive systems for the operation of complex technical objects on the basis of detailed modeling of the work and the interaction of their individual components with the help of heavy finite-element packets for numerical analysis. The fact that predictive calculations with their help can be carried out in real time
or even ahead of the time of the onset of a real event. And this gives additional time for the operator to analyze the situation and make the right decision.

This work presents the elements of such a forecasting system for a physical object - an electrically-heated insulated pipeline (an experimental stand), the mathematical type and its numerical implementation package ANSYS, a system for automated collection and processing in real time information (for control of heat stand state and sufficiency of the reproduction of its thermal domain in ANSYS).

2. Description of the laboratory stand
Experimental studies have been conducted in educational and research laboratory “Heat pump systems” departments of Theoretical bases heating engineering on the experimental stand. Full-scale heat-insulated pipeline equipped with industrial cable system of electric heating of the enterprise “Special systems and technologies”.

2.1. Technical parameters
In figure 1, the schematic diagram of the stand integrated with the peak closer ‘9’ connected with a ground heat exchanger is shown.
- A diameter of the warmed pipe – 40 [mm];
- Length of a horizontal part of the pipeline – 8.57 [m];
- The maximum power of a heating system – 1.9 [kW];
- Power output in heating sections at a temperature of 40 °C - 53 [W/m];
- Heat-insulating layer--foamed rubber with a thickness of 35 [mm].

![Figure 1. The stand with the heat pump.
I- Heat Pump installation; II- The peak closer; III- Boiler.](image)

2.2. Control system of cable heating of the stand
Control system of cable heating of the stand is shown in figure 2, containing the following buttons:
- SF1: introductory circuit breaker;
- SF2: a device of protection shutdown saves the person from electric shock;
- S01: the switch connected with temperature regulator;
- A2: temperature regulator;
- HLG1: shows that the power is connected;
- HLG2: shows that the heating is connected;
- HLG3: shows that the alarm is connected;
- SF3, SF4, SF5: four sectors to provide different power outputs through the pipe section ‘figure 3’.

Figure 4 shows the outer shape of the flat heating tape FSS.

Figure 2. Control system of cable heated pipe.

Figure 3. Scheme of an arrangement of heating cables and thermocouples on a pipe surface.

N6- water temperature.

2.3. Temperature characteristics of the Cable 60FSS

The nominal heat dissipation of the 60FSS tape at 230 V on insulated metal pipes can be represented by the equation:

\[ q = -0.3167t + 70 \] (1)

Where q – power output “heat release” (W/m) and t – temperature of a cable (°C).

2.4. Measuring system

The experimental setup has been equipped with advanced equipment which have computer interface to monitor and control the temperature in 12 different points where it is necessary to take temperature at the same time in several points for assessment of distribution of temperature with the help of the TM12 thermo-measuring instrument, pressure (two sensors input / output) by means of PD100-pressure sensor DI and a liquid consumption in real time with help of computerized system (ADC) for collecting and processing information ‘figure 5’.
3. Solution with using the ANSYS program

The practically important problem which is the heating of the pipeline filled with a coolant (water) in the event of a stoppage of pumping has been experimentally and numerically investigated.

3.1. The mathematical model in ANSYS

The area of interest - the section of the pipeline ‘figure 3 and 8’ has been divided into five sub-areas: 1 - water; 2 - pipe wall; 3 - thermal insulation; 4 - cable insulation and 5 - core of cable. In two zones, the steel pipe wall (zone 2) and cable insulation (zone 4), the dependence of the coefficient of thermal conductivity on the temperature is taken into account. In the cable core (zone 5), the heat release due to the passage of electric current has been modeled as an internal volumetric heat source $q_v$ as a function of temperature using the equation (1) according to the geometric dimensions of the cable. The convective boundary condition (the coefficient of heat transfer) on the surface of the thermal insulation layer in contact with the atmosphere has not been determined. However, it has been refined iteratively in the solution process. Thus, it was necessary to solve the 2-D problem of varying heat conductivity (zone 2) in the region of complex geometry in the presence of zones with a temperature-dependent thermal conductivity and zones with a temperature-dependent internal energy release. The construction of an analytical solution of such a problem for calculating the temperature field causes obvious difficulties and it is natural to perform a computational experiment.

The differential equation of unsteady heat conductivity in a two-dimensional formulation can be written as follows:

$$\rho C_p \frac{\partial t}{\partial t} = \frac{\partial}{\partial x} (\lambda \frac{\partial t}{\partial x}) + \frac{\partial}{\partial y} (\lambda \frac{\partial t}{\partial y}) + q_v$$

(2)

Where $\rho$ – material density, kg/m$^3$; $C_p$ – the specific heat of material, J/(kg.k); $\lambda$ – coefficient of heat conductivity, W/(m.k) and $q_v$ – the power of internal sources, W/m$^3$.

The solution of the equation (2) is carried out in ANSYS by the finite element method.

3.2. Stages of the solution of a task in ANSYS

- Creation of the problem geometry for the object under study:
  Initially, at this stage, it is necessary to determine the areas with their dimensions, thermal and physical properties of the materials and the way of creating them directly in ANSYS graphical editor or import the geometry created in the CAD package ‘figure 6 and 7’ into the ANSYS calculation module ‘figure 8’.
  In our case, the area of interest has been originally created using AutoCAD software, and then it was exported to ANSYS software in the "IGES" format ‘figure 6 and 7’.

Figure 4. Nominal power output against pipe temperature.

Figure 5. Computerized system to collect and process the information integrated with TM-12 thermo-measuring instrument.
Mesh Generation:
Initially, the built-in grid generator is used. At the nodes of the grid, solution of the equation (2) is to be obtained. The number of grid nodes determines the number of equations that ANSYS must solve ‘figure 9’.

Setting of initial and boundary conditions, physical properties of the studied object and calculation parameters:
The initial and boundary conditions are illustrated in ‘figure 10 and 11’ respectively. In figure 11 the boundary conditions of the third kind are given on the outer surface of thermal insulation by the coefficient of heat transfer and the temperature of the liquid.

Carrying out calculation (numerical solution):
At this stage, the type of built-in solver, the time steps, the physical duration of the process and the number of iterations is chosen to solve the non-linear problem ‘figure 12’.

Initialization of calculation:
The system of algebraic equations is solved ‘figure 13’. 
4. Determining of the average heat transfer coefficient from the thermal insulation surface to the atmosphere

Initially, the heat transfer coefficient for free convection of air around a horizontal pipe is estimated from the experimental value of temperature, then this value is used as a boundary condition in ANSYS for calculating the temperature field in the pipeline by solving the unsteady heat conduction problem of a multilayer cylindrical wall. Further, the surface temperature of the thermal insulation of the pipeline $t_{av}$ from the ANSYS solution is used to correct the heat transfer coefficient and the calculation is repeated in ANSYS. According to the tables of thermal and physical properties of air [1], we can find the coefficient of kinematic viscosity of the environment (atmospheric air), the coefficient of thermal conductivity of the environment, the Prandtl number, at $\Delta t = t_{av} - t_{a}$ (air temperature), by the external diameter of the thermal insulation, we find Grashof number $Gr$.

By means of Grashof and Prandtl numbers we find the Rayleigh number. Furthermore, we find the Nusselt number [2] with $Ra_d = 10^4 \ldots 10^7$:

$$Nu = 0.5Ra_d^{0.25}$$  \hspace{1cm} (3)

Further, we determine the average coefficient of heat transfer in the case of free convection of air on a horizontal cylinder by the formula:

$$\alpha = \frac{Nu \lambda}{d}$$  \hspace{1cm} (4)
5. Results
Figure 14 shows the results of an experimental study of the heated water in thermally-insulated pipeline equipped with a self-regulating 60FSS cable. The cable heats up to a temperature of 112.87°C ‘figure 14’ during a period of 30 minutes with the maximum temperature of the heated water inside the tube being 64.54°C.

![Figure 14. Change of sensors temperature over time.](image)

The developed numerical model of the experimental setup for calculation in ANSYS, presented above, has made it possible to perform a computational process of the same system, the temperature distribution at the final instant of time is shown in ‘figure 15’. The calculation time on a laptop with an Intel Core i7 processor was about 17 minutes, which is almost half the time of the full-scale experiment.

Figure 16 shows the dynamics of temperature changes in the process of full-scale and computational experiment. A good match was found for 6 of the 10 temperature sensors.

6. Conclusion
The numerical model of installation is developed for calculation in ANSYS including zones: (water, pipe wall, insulation, cable cores with the q, variables, electrical cable insulation).

![Figure 15. The domain of temperature in the pipe section at the end of the experiment.](image)
Comparison of the dynamics of experimental results with the numerical calculation by mean of ANSYS showed a good agreement, which allows us to recommend the developed numerical model for calculating the thermal states of similar systems.

7. References
[1] Aleksandrov A, Orlov K, and Ochkov V 2009 *Thermophysical properties of thermal power engineering working substances* (M.: Izdatelskiy dom MEI) p 224.
[2] Tsvetkov F, Kerimov R and Velichko V 2008 *Book of problems on heat-mass transfer* 2nd ed. updated and revised (Moscow: MPEI Publishing House) pp 196.