Calculation of coaxial shunts superheat temperature

E V Bedareva ¹, P S Marinushkin ² and L I Khudonogova ¹
¹ Tomsk Polytechnic University, Tomsk, Russia
² Siberian Federal University, Krasnoyarsk, Russia

E-mail: bedareva@tpu.ru

Abstract
Coaxial shunt temperature field distribution has been obtained and analyzed using software package ANSYS. The main stages of 3D modeling of coaxial shunt temperature field in ANSYS are considered in the paper. It is determined that coaxial shunt temperature field distribution is nonuniform because of the material physical properties and free convection. The difference between theoretical temperature cylinder (60 ºC) and experimentally observed temperature value (49 ºC) of resistive pipe is caused by the fact that free convection and heat emission were not taken into account when calculating.

1. Introduction
Coaxial shunts are widely used as part of electric energy control devices and electrical power system test equipment, for precision AC and DC currents measurements over the wide dynamic and frequency ranges [1-5].

The main factor that determines the size of resistive element is load-carrying capacity. When choosing the overall dimensions, it is necessary to take into account resistive element superheat temperature [4], representing the difference between surface temperature of resistive element and ambient temperature.

The purpose of this paper is to carry out a thermal field modeling and calculate the value of superheat temperature of shunt prototype. This research will allow to determine critical operating modes and to find the maximum possible pulse current which will not lead to shunt destruction.

2. Coaxial shunt design
Figure 1 shows the structure of a coaxial shunt, consisting of an inner resistive cylinder made of manganin and the outer cylinder made of copper. Currents flow through the inner and outer cylinders in opposite directions. Thus magnetic field cancellation is achieved. The structure cooled by natural convection [1].

Simulations were performed for the coaxial shunt with the following parameters: resistive cylinder thickness – 1 mm; length – 90 mm; inner radius – 10 mm; the gap between cylinders – 1 mm; rated current – 1 kA; pulse duration – 1 s; transient period – 2,79 mks; superheat temperature – 40 ºC. The inner cylinder is made of manganin, external – of copper, current and potential leads – brass.

3. Coaxial shunt thermal field modeling
Modeling of the shunt temperature field was carried out using finite element method (FEM) in software package ANSYS Mechanical.

Modeling process in ANSYS Mechanical consists of the following stages:
1) Importing a model created in a computer-aided design (CAD) system, or creating a solid model within ANSYS;
2) Gluing the parts;
3) Choosing the type of elements;
4) Specifying material properties;
5) Generating the mesh;
6) Applying boundary conditions and loads;
7) Defining dynamic analysis options;
8) Obtaining the solution;
9) Reviewing results.

Figure 1. Coaxial shunt design

Geometrical parameters were calculated by means of the express calculator [6]. The solid model of a shunt was developed in Inventor Professional software package. Additional volumes representing areas filled with air were introduced into model of the shunt to account the heat exchange between parts.

To import the model from file (in SAT format) it is necessary to do the following operation: Utility>Menu > File > Import > SAT.

After importing the model has wireframe representation. To fix it and generate volumes the following operation should be done: Utility > PlotCtrls > Style > Solid Model Facets > Solid Model Facets > Normal Faceting.

For further modeling it is required to create common surfaces for neighbouring parts, which can be provide by gluing operation: Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes > Pick All. Contact phenomena between parts doesn’t take into account.

Mesh generation is one of the most important stages for finding the solution. The choice of element type affects the set of degree of freedom, which, in his turn, determine the analysis type: structural, thermal, magnetic, electric, etc.

In this application, for 3D analysis, SOLID70 element for modeling air gapees and SOLID69 element for modeling metals are chosen. (Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Thermal Mass > Solid > Scalar Tet 70, then Coupled Field > Scalar Tet 69).

For all materials, including in the prototype model, four physical properties are to be specified: thermal conductivity, thermal capacity, density, and resistivity. Inner pipe is made from manganin, and the outer one is made from copper. For current and voltage terminals brass is chosen. The specific material properties are defined by executing the command set: Main Menu > Preprocessor > Material Props > Material Models > required properties choosing.

The next step in the modeling process is the mesh generation. In this particular case tetrahedral shaped elements mesh were used. For meshing operations a meshing feature called smart element sizing ("SmartSize") was applied. It determined element size of 10 for both manganin and copper.
pipes, and size of 6 for the other parts of the prototype. The reason of the different element sizes for different parts is that pipes are the most essential part for finding the solution and obtaining the temperature field distribution. Mesh generation is performed by the following operations: Main Menu > Preprocessor > Meshing > MeshTool > set the “Element type number” > choose the material in accordance with the elements type > set the value of SmartSize parameter > set the parameter “Shape” = “Tet” (tetrahedral mesh) > set the parameter “Shape” = “Free” (Free meshing) > choose the part to be meshed in “Mesh Volumes” window.

The next stage is applying loads and boundary conditions. In the given case loads on elements are represented by:
1) Zero potential;
2) Current value of 1 kA;
3) Free convection.

Zero potential (A) and current (B) are applied on current terminals surfaces as shown in figure 2. Free convection (heat-transfer coefficient is 30 W/m²·ºC, ambient temperature is 20 ºC) is applied on surfaces set off in turquoise, i.e. on all external surfaces.

![Figure 2. Applying loads and boundary conditions](image)

Process of setting the input current pulse consists of three load steps. The first one is characterized by the following parameters: initial time is 0.1 s and initial temperature is 20 ºC. For the second step transient period is 2.79 mks, pulse duration is 1 s, and current value is 1 kA. In the third step current amplitude is 0 A, time to exposure completion is 5 s.

After all preprocessing stages, the performed task can be solved. To obtain the solution the following operation should be done: Main Menu > Solution > Solve > From LS Files. The order of load steps is specified in the options window.

In the postprocessing stage, obtained results visualization is implemented by the following commands: Main Menu > General Postproc > PlotResults > Contour Plot > Nodal Solu > “DOF solution” > “Temperature”.

Figure 3 shows coaxial shunt temperature field distribution for the period of one second current pulse exposure.
According to figure 3, when large-amplitude pulse current flows through the shunt, maximum superheat temperature occurs on the inner cylinder, decreasing towards the ends. Non-uniform thermal distribution of resistive element takes place because of the physical properties of materials, which current and voltage terminals are made from. The maximum temperature value is 49.2 °C.

Plot in figure 4, demonstrating temperature-time dependence for node 1 (figure 3), allows to estimate dynamic temperature changing of the resistive pipe in different time points. It is relevant to note that temperature-time dependence for other nodes of inner pipe is analogous to the one represented in figure 4.

When current pulse is stopped, shunt superheat temperature begins to decrease due to heat emission and heat exchange between other parts.

Figure 3. Coaxial shunt thermal field distribution at time $t = 1$ s

Figure 4. Resistive pipe temperature changing in time
4. Conclusion
In this paper coaxial shunt temperature field distribution has been obtained and analyzed using ANSYS. From the results obtained, the following conclusions can be made:

1) The theoretical temperature of a resistive cylinder (60 °C) given by the calculations doesn't coincide with experimentally observed temperature value (49.2 °C) because the theoretical calculations do not take into account natural convection and heat emission.

2) Ohmic shunt resistance does not change when temperature increases, hence, shunt behaves as a linear stable system.

3) Current pulse, effecting on the shunt, is not critical, so current amplitude can be therefore increased, and pulse duration can be extended as well.

4) Maximum superheat temperature occurs in the center of the inner pipe that caused by free convection and physical properties of the materials used in the model.

References
[1] Bedareva E, Tsymbalist E, Muravyov S and Baranov P 2013 Influence of methods of potential terminal connection on dynamic behavior of coaxial shunts Bulletin of Tomsk Polytechnic University 322 pp 159–164
[2] Muravyov S, Borikov V and Natalinova N 2009 A Computer system: measurement of welding surge current Measurement and Control 42 pp 44–47
[3] Kawamura T, Haginomori E, Goda Y and Nakamoto T 2007 Recent developments on high current measurement using current shunt Transactions on electrical and electronic engineering 2 pp 516–522
[4] Zarevich A, Muravyov S and Bedareva E 2013 Digital signal processing in pulse method for measuring frequency response function of high-current shunt 19th IMEKO TC4 Symposium pp 496–500
[5] Muravyov S, Borikov V and Kaysanov S 2006 Computer system for measurement of welding process parameters 18th IMEKO World Congress 2 pp 77–80
[6] Borikov V 2010 Computer-aided calculation of electrical and design parameters of coaxial shunts in LabVIEW professional development system Prybory 3 pp 42–46