Calculation of temperature field of coiled wire using EFM

Zhongjun Shu[1], *Wei* Shen[2], *Qiang* Li[1], *Minghao* Fan[2], and Jiaqing Zhang[2]

[1]Chinese People's Armed Police Academy, Department of Fire Protection Engineering, 065000 Langfang, China
[2] State Grid Anhui Electric Power Research Institute, 230022 Hefei, China

Abstract. Provided a heat transfer model of coiled wire method. Based on the method, a software of EFM (ANSYS) was used to calculate the temperature field of coiled wire. Comparisons between the experimental of RVS coiled wire and numerical results indicated the effectiveness of the method utilized. The simulation method based on EFM proved to be useful for the fire risk assessment of coiled wire.

The non-standard operations of wire are important cause of electrical circuit fault, which in turn will cause electrical fire. There is exist a problem that wires are usually tightly coiled or stacked when they are used in our daily life due to easy to use and manageable. The problem is even more common in construction site and outdoor venue. In coiled states, the continuous current rating of wire will reduced caused by heat accumulation and poor heat dissipation. This situation, therefore, may cause the wires can not meet the ability to supply power loads, which wires could have to meet. Results show that overloading will accelerate insulation aging of wire, cause deterioration of the insulation, and shorten the life span of wire. In the worst of conditions, the wire fault is possible to cause fire. In order to prevent electrical fires and reduce the possibility of fire caused by wire faults, it is necessary to study the calculation method of temperature field of coiled wire.

Thermal circuit method within IEC 60287 is often used to analyze the temperature field of wire. The method only works with calcuating the temperature of wire core during steady-state conditions, rather than prediction of temperature field of wire at different time. Compare to thermal circuit method, numerical method offers an approach to predicting the temperature field of coiled wire. The heat transfer models were established in this paper. Based on these models, we studied the finite element method (FEM)of temperature field of coiled and presented the fire risk assessment method of coiled wire based on the FEM.

1 The heat transfer models of coiled wire

To simplify the heat transfer models of coiled wire, these are assumed:

(1) the stacking condition of coiled wire is ideal form, that means wires keep tangent to other wires and ground, and the deformation of wire caused by the pressure or heat expansion is ignored;

(2) ignore the energy transmits in coiling direction, the temperature gradient of coiled wire only exists in the coiling radius direction and vertical radius direction, that is to say, there are energy transmits in these two directions;

(3) ignore the effect of temperture on the resistivity of conductor of wire, the heat of live wire can be calculated according to Ohm's Law;

(4) ignore the heat absorbance of wire insulation in the process of dissociation, physical and chemical properties of insulation materials remain the same before the insulation failure;

(5) ignore the effect of interfacial thermal contact resistance, insulation material and conductor are good contact;

(6) only heat conduction between the wire and the air in the pore;

(7) multi-core wires are regarded as single core wires with same cross-sectional area;

(8) the ambient temperatures keep constant and the fluid properties of air are unchanged.

Based on the above assumptions, the heat transfer models of coiled wire are established, as shown in Figure 1.

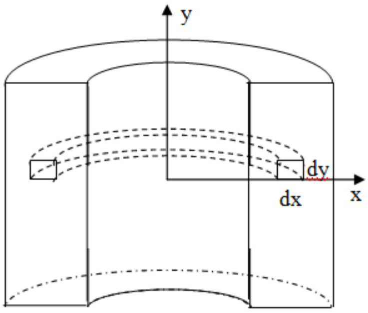


Fig. 1. 2-dimensional thermal conductivity model of volume element is expressed in Cartesian coordinates.

In the sight of basic principle on conductions of heat, Fourier heat conduction differential equation of the volume element of heat conductor is founded. Net heat of the volume element per unit time is shown as formula (1).

$$\mathrm{d}Q = 2K\pi\left(\frac{\partial T}{\partial x} + x\frac{\partial^2 T}{\partial x^2}\right)\mathrm{d}x\mathrm{d}y + 2\pi K x\frac{\partial^2 T}{\partial y^2}\mathrm{d}x\mathrm{d}y \qquad (1)$$

where K is thermal conductivity, W/(m·K). T is tempertaure, K.

Heat generated by the volume element per unit time is shown as formula (2).

$$\mathrm{d}Q_g = q[\pi(x+\mathrm{d}x)^2 - \pi x^2]\mathrm{d}y \approx 2\pi q x\mathrm{d}x\mathrm{d}y \qquad (2)$$

where Qg is the heat generated by the volume element, J. q is the heat released from the unit volume per unit time, $\mathrm{J}/(\mathrm{s}\cdot m^3)$.

Based on assumption (4), the heat absorbance of wire insulation in the process of dissociation is ignored. Therefore, the conductor with power can be considered as heat source with uniform distribution. On principle of Joule's law, the heat released from the unit volume per unit time is shown as formula (3).

$$q = I^2 r = I^2\frac{R}{LS} = I^2\frac{\rho L}{S\cdot LS} = I^2\frac{\rho}{S^2} \qquad (3)$$

where I is current, A. r is volume resistivity, Ω/m^3. R is resistance, Ω. L is length, m. S is cross-sectional area, m2. ρ is resistivity, Ω/m.

With rising temperature of the volume element caused by absorbing heat, the change of internal energy per unit time is shown as formula (4).

$$\mathrm{d}E = C\mu[\pi(x+\mathrm{d}x)^2 - \pi x^2]\mathrm{d}y\frac{\partial T}{\partial t} \approx 2\pi C\mu\, x\mathrm{d}x\mathrm{d}y\frac{\partial T}{\partial t} \qquad (4)$$

where C is specific heat capacity, $\mathrm{J}/(\mathrm{kg}\cdot\mathrm{K})$. μ is density, kg/m3. t is time, s.

Energy conservation is shown as formula (5).

$$\mathrm{d}Q + \mathrm{d}Q_g = \mathrm{d}E \qquad (5)$$

Appling formula (1), formula (2) and formula (4) into formula (5), the finite element equations of temperature field is gotten, that is shown as formula (6).

$$K\left(\frac{1}{x}\frac{\partial T}{\partial x} + \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + q = C\mu\frac{\partial T}{\partial t} \qquad (6)$$

The finite element equations of temperature field of coiled wire cross section is shown as formula (7) and formula (8).

$$K\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + q = C\mu\frac{\partial T}{\partial t} \qquad (7)$$

$$q = \begin{cases} 0, & \text{without heat source (Insulation area)} \\ I^2\dfrac{\rho}{S^2}, & \text{with heat source (Copper core area)} \end{cases} \qquad (8)$$

The boundary conditions of coiled wire are needed for solving the heat conduction differential equations. Three kind of boundary conditions are typically used in heat transfer theory. The first is known boundary temperature, and the second is known heat flux, and the third is known boundary condition of convection. The heat convection between the air and the insulation material of wire is the third kind, therefore, it can be expressed as formula (9).

$$\begin{cases} K\left(\dfrac{\partial^2 T}{\partial x^2} + \dfrac{\partial^2 T}{\partial y^2}\right) + q = C\mu\dfrac{\partial T}{\partial t} \\ -K\dfrac{\partial T}{\partial n}\mid_{\Gamma} = h(T - T_f)\mid_{\Gamma} \end{cases} \qquad (9)$$

where h is convective heat-transfer coefficient, $\mathrm{W}/(\mathrm{m}^2\cdot\mathrm{K})$.

2 Numerical simulation of temperature field based on FEM

2.1 The finite element model of coiled wire

For processing FEM, the computer assistant analysis is used after dispersing the differential equation. ANSYS, the finite element analysis software, is typically applied to handle FEM. We take an arbitrary cross section, which along radical direction of coiled wire, to establish a 2-dimension model for the temperature field. Based on the assumptions, the vertical cross section of wire can be divided into two concentric circles when we take the single wire as the basis unit. Inner circle is the conductor of wire, torus is the insulating material of wire. The physical model of the cross section of the coiled wire, as shown in Figure 2, is established by copy the basis unit along x-axis and y-axis. On the basis of the physical model, the division of quadrilateral mesh mode is adopted to partition meshes of 2dimension model directly. The coiled wire physical model is divided based on the relative distance to conductor by the rule: the closer distance from the conductor, the more dense with mesh, as shown in Figure 3.

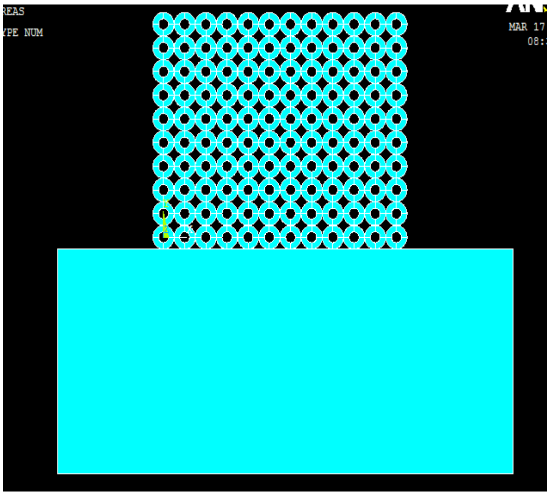


Fig. 2. Physical model of the cross section of the coiled wire.

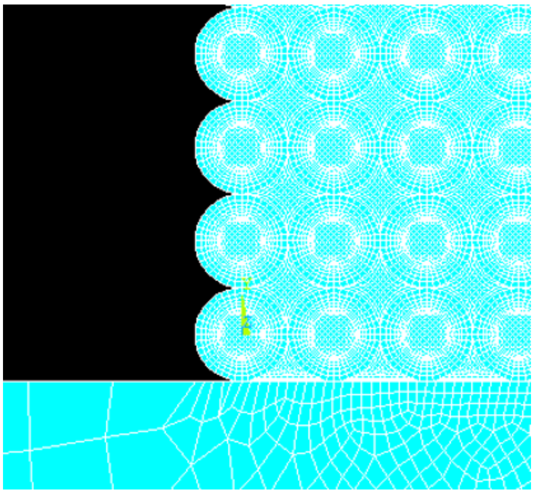


Fig. 3. Locally thickened is deployed in the flow field mesh division.

The properties of wire conductor and insulation material can be set in ANSYS. Heat producing rate is set for simulating the power-up state of wire that can be used to achieve the simulating of temperature field of the coiled wire. With the coiling 60 laps of RVS 2×2.5 wire, for example, the temperature field of coiled wire is simulated by ANSYS. The parameters of RVS 2×2.5 sample are listed in Table 1.

Table 1. Parameters of RVS 2×2.5 Sample

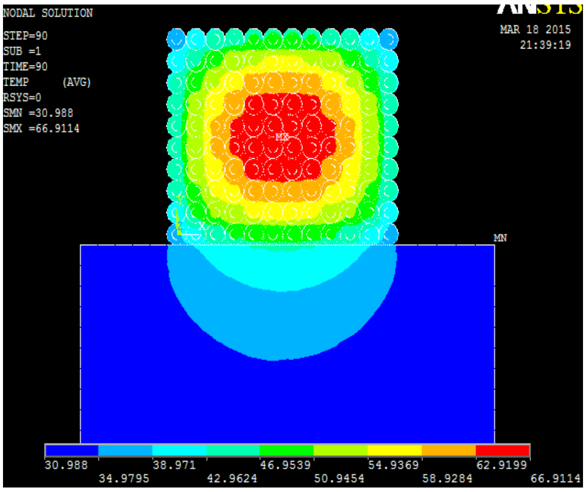
Items	Parameter /Unit	Value
Wrie	Diameter /mm	3.72
Copper core	Diameter /mm	1.72
	Thermal conductivity / W˙m-1˙K-1	387
	Specific heat capacity /J˙kg-1˙℃-1	0.39×103
	Density /kg˙m-3	8 940
	Resistivity /Ω˙mm2˙m-1	0.018
PVC insulation material	Thickness /mm	1.00
	Thermal conductivity / W˙m-1˙K-1	0.25
	Specific heat capacity /J˙kg-1˙℃-1	1×103
	Density /kg˙m-3	1 400
Ground (concrete)	Thermal conductivity / W˙m-1˙K-1	0.78
	Specific heat capacity /J˙kg-1˙℃-1	1.5
	Density /kg˙m-3	2 300
Air	Thermal conductivity / W˙m-1˙K-1	0.026
	Specific heat capacity /J˙kg-1˙℃-1	1.04×103
	Convective heat-transfer coefficient / W˙m-2˙K-1	17.5
	Density /kg˙m-3	1.1

2.2 Experiment of heating coiled wire

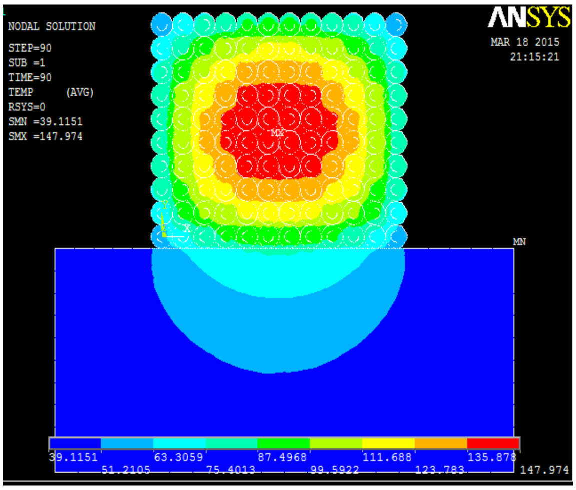
RVS 2×2.5 wire with coiling 60 laps is used in the experiment. The effect of load on temperture field of coiled wire is studied in this research. There are two different loads listed as follows: 1 200 W and 2 500 W. Thermocouples are used to measure the temperature of each part of the coiled wire, including the outer wall, inner wall, underside, insider of parallel wire and center of a circle of the coiled wire.

2.3 Result analysis

Temperature fields of coiled wire, which calcuated by ANSYS, are shown in Figure 4. Convective heat-transfer coefficient is 17.5 W·m-2·K-1. The load of Figure 4(a) and Figure 4(b) are 1 200 W and 2 500 W respectively



(a) Connect 1 200 W electrical appliances



（b）Connect 2 500 W electrical appliances

Fig. 4. Temperature fields of RVS 2×2.5 wire with coiling 60 laps in the 90th minute.

In Figure 4, the isotherm present circle line. The highest temperature is at the center of the vertical cross section of coiled wire, and the temperature decreases gradually from the center outward. 147.974 ℃ and 66.911 ℃ are the maximum temperature when the load are 2 500 W and 1 200 W respectively.

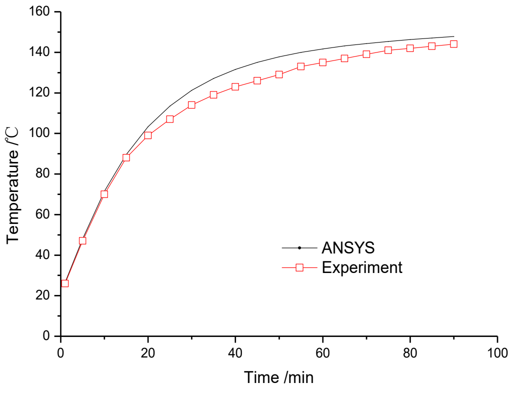
The experimental results and the ANSYS calculation results in 90 min after power on are listed in Table 2.

Table 2. The Temperature of Each Part of Coiled Wire
Unit: °C

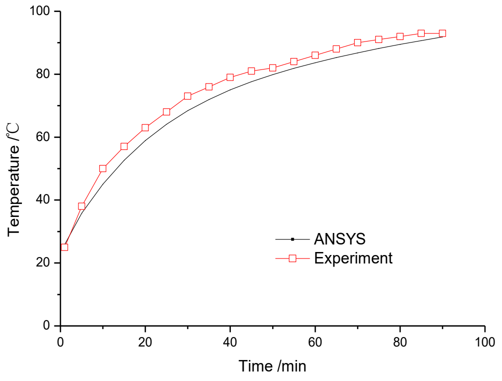
Load	Measuring part	Experimental measurement	ANSYS
1 200 W	Insider of parallel wire	66	66.86
	Inner/outer wall	54/47	45.66
2 500 W	Insider of parallel wire	144	147.81
	Inner/outer wall	85/72	80.55
	Underside	93	91.86

Table 2 shows that, in 1 200 W scene, the result of insider of parallel wire temperature calculated by ANSYS software agree with the experimental result, and the ANSYS result of wall central point temperature is slightly lower than the temperature of inner wall which gets measured on the same conditions. For the 2 500 W scene, the ANSYS result of insider of parallel wire temperature is slightly above experimental result, and the temperature of wall central point calculated by ANSYS lies between the temperatures of inner wall and outer wall which measured experimentally. The ANSYS result of underside temperature is slightly lower than the experimental result.

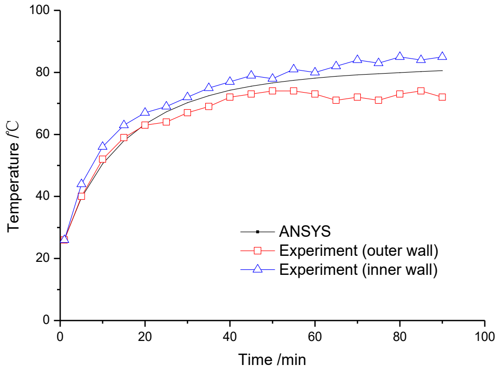
Figure 5 shows temperature-time curve of each part of coiled wire in load 2 500 W scene.



(a) Insider of parallel wire



(b) Underside



(c) Wall

Fig. 5. Temperature-time curve of each part of coiled wire (coiling 60 laps, P=2 500 W).

Figure 5(a) shows the temperature inside the coiled wire. The comparison between the temperature calculated by ANSYS and experimental result is showed below. In the first 15 min loaded, the two temperature curves are coincident, the temperatures exceeds the softening temperature of PVC insulation. After 15 min, the temperature difference between the two curves is slightly, and the temperature difference first increases and then decreases with the increase of the time. Such deviation may be caused by the change of the physical properties, thermal parameters of the insulation material and electrical resistivity of the copper conductor in the process of temperature rise.

The insulation material PVC is amorphous polymers. It is at glass state when the temperature is low, and it will only occur very small deformation under the action of external force; when the heating temperature was increased to 80 ℃ and reached the glass transition temperature, PVC changed into rubbery state, whose specific heat capacity, expansion coefficient, viscosity, free volume and elastic modulus are mutated. The specific heat capacity of PVC increased after it changed into rubbery state, and the heat absorption increased with temperature increased. At the same time, the resistivity increased with increasing of the conductor temperature, and the heat generated by the wire in unit time increase. These factors work together, resulting in the deviation between the calculated values and the measured values after 15 min loaded.

Figure 5(b) shows the temperature underside the coiled wire. The comparison between the temperature calculated by ANSYS and experimental result is showed below. The trend of the two temperature curves is basically the same, the simulated temperature is slightly lower than the experimental temperature. This may be caused by the gravity which made the wire at the bottom more closely and then the heat transfer to the ground increases. The thermal parameters of ground material used in ANSYS are regard as the concrete material's and there is a little difference. Therefore, the simulated bottom temperature is slightly higher than the experimental bottom temperature.

Figure 5(b) shows the temperature underside the coiled wire. The comparison between the temperature calculated by ANSYS and experimental result is showed below. The simulated temperature curve is much smoother than the experimental temperature curve. In the first 20 min loaded, the two temperature curves are coincident. After 20 min, the simulated temperature curve is between the measured inside and outside wall temperature curves. At the same time, the simulated temperature at the bottom surface is higher than that in the middle point of the outer wall and lower than that of the inner wall temperature. This deviation is caused that the setting of convection heat transfer coefficient used in ANSYS is constant. In the experiment, the temperature of coiled wire increased with the increasing loaded time, and the convective heat transfer coefficient increased with the increasing of air temperature which is very sensitive to the environmental ventilation.

The comparison shows that the temperature curves simulated by ANSYS are close to the experimental temperature curves. This shows that the temperature in different parts of the coiled wire simulated by using ANSYS to establish finite element model is accurate, and the simulation can meet the actual requirement of judging the fire risk of the coiled wire.

4 Fire Risk Analysis

To judge the fire risk of coiled wire for a long time loaded, we used ANSYS to simulate the coiled wire temperature rise under different conditions which have different coiled ring number, load heat production and other heat transfer parameters. Figure 6 shows temperature-time curve of RVS 2×2.5 wire with coiling 60,70,80 laps in load 2 500 W scene.

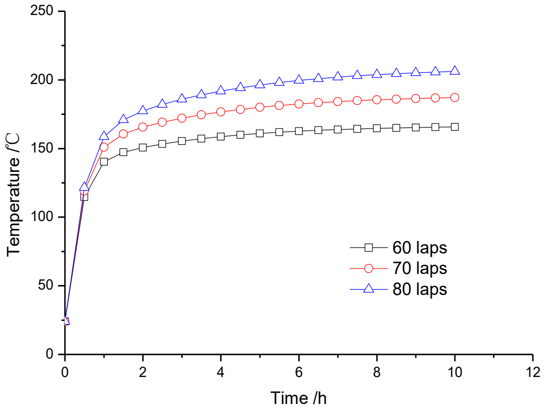


Fig. 6. Temperature-time curve of RVS 2×2.5 wire with coiling 60,70,80 laps (P=2 500 W).

In the first 2 hours loaded, the temperature of RVS coiled wire changes significantly, then the temperature risk slowed markedly which approximately reach steady-state temperature rise. The temperature of the wire loaded for 10 h is considered as the steady state temperature. The steady-state temperature of RVS wire with coiling 60 laps is 165.71℃ which is higher than the ignition point of normal combustible material, the coiled wire under this condition may ignite combustible material with low ignition point. The steady-state temperatures of RVS wire with coiling 70 and 80 laps are 187.03 ℃ and 206.15 ℃ which close to or reached the pyrolysis temperature of PVC insulation, the coiled wire under this condition will caused fire accident by metal wire exposed.

5 Conclusion

Based on establishment of heat transfer models of coiled wire, FEM was applied to simulating the temperature field of coiled wire in this paper. The main conclusions were as follows:

(1) By comparison with experimental results of heating coiled wire (RVS2×2.5), the computational results of each part temperature of coiled wire calcuated by ANSYS are in good agreement with experimental results on the same conditions. That proved that FEM is available to calculate the temperature field of coiled wire.

(2) The results demonstrate that the wire in coiling state will result in fire hazards caused by great heat or insulation failure even with normal load, and the fire hazard rises as just because of coiling number of coiled wire increase.

References

1. J. Li, Q. Li, 11th academic annual conference held by China Association for Science and Technology. Chongqing: China Association for Science and Technology. (2009)
2. S.K. Li, Building Electricity **1**, (2011)
3. Z.Y. Wei, G.H. Cai, Journal Of Chinese People's Armed Police Force Academ, **25**, 6 (2009)
4. IEC 60287-1-1, IEC Publication, (1994)
5. IEC 60287-1-2, IEC Publication, (1994)
6. Z.C. Xu, Heilongjiang Electric Power **5**, (2013)
7. Y.Y Wang, R.G. Chen, W.G. Chen, High Voltage Engineering **36**, 12 (2009)
8. Y.R. Li, S.Y. Li, Beijing, Science Press, (2012)