NEW FEM MODEL FOR THERMAL ANALYSIS OF MEDIUM VOLTAGE FUSES

Esther TORRES Angel Javier MAZÓN Elvira FERNÁNDEZ Inmaculada ZAMORA
University of the Basque Country (UPV/EHU) – Spain
esther.torresi@ehu.es javier.mazon@ehu.es elvira.fernandezh@ehu.es inmaculada.zamora@ehu.es

Juan Carlos PÉREZ
MESA – Schneider Electric - Spain
juan-carlos.perez-quesada@es.schneider-electric.com

ABSTRACT

Electric fuses are protection devices broadly used in distribution networks. They are designed to interrupt the circulating current by means of the melting of one of their components when excessive currents flow through them during a period of time. The knowledge of the thermal behaviour of the fuse is not only essential to define its operation characteristics, but also to prevent any damage to other equipment because of over temperature. The object of the work presented in this paper is to provide fuse manufacturers with a model, based on the finite element method, which allows them to know the thermal behaviour of medium voltage current limiting fuses under different conditions of current flow and ambient temperature, and so to reduce the need for testing.

INTRODUCTION

Fuses are among the best known electrical devices because of their fast and reliable operation in addition to their low price, in comparison with other breaking devices. They are used for a wide range of voltage levels: from low-voltage levels (<1kV) in domestic uses to high-voltage values (≥1kV) in electrical distribution networks.

This work is focused on medium-voltage fuses, and more precisely, on back-up current-limiting fuses used for protection of medium voltage networks and electrical equipment. Current-limiting fuses are composed of copper or silver conductor tapes (fuse elements), wounded on a star-core and connected to two conductor pieces or end caps. Both the fuse elements and the star-core are inside of a ceramic tube full of quartz sand. Besides, frequently, fuses are also equipped with a striker, which provides visual indication of the fuse operation (Fig. 1).

Fuse standards [1] limit temperature values achieved by contact pieces and other materials in the fuse, under the rated current. This makes necessary the development of prototypes as well as testing and evaluation of them to check the compliance with temperature limits.

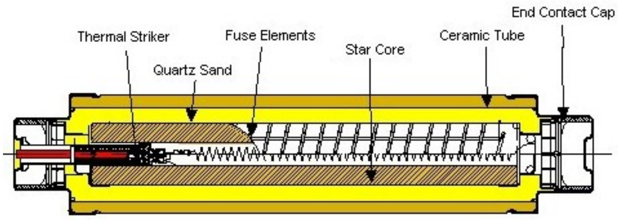

Fig. 1. Longitudinal section of a fuse

The thermal problem in electric fuses has been tackled by different authors ([2]…[9]), who have focused their analysis on the fuse element ([2], [3]), or larger portions of the fuse ([4]…[9]). Some authors have chosen an analytical approach to solve the thermal problem ([2], [4]). Other authors use techniques as the finite element method ([7], [9]), finite differences ([3], [8]) or others ([5], [6]), due to the complexity of solving the heat equation or to make possible the analysis of a bigger portion of the fuse.

In any case, whichever technique is applied, a simulation process includes both the design of the model and the experimentation processes to study its behaviour. On this way, simulation allows to experiment with the model rather than with the real system, which is of great help for designers, because the equipment under study can be examined, doing changes in the model without having to carry out physical changes.

The purpose of this paper is to provide fuse manufacturers with a model, based on the finite element method. This tool allows to know the thermal behaviour of a complete medium voltage current limiting fuse under different conditions of current flow and ambient temperature, and so to check the compliance with standards. The models presented in this paper represent an advance with regard to those presented in [10], as they model the fuse in a more real way.

MODELLING AND SIMULATION

In this section two different models developed to reproduce the thermal behaviour of the fuse are described and compared.

These models have been designed with the Solid Edge software tool and the finite element analysis has been performed using the Cosmos DesignStar finite element analysis package. The most difficult problem in the modelling stage has been the modelling of the fuse elements because of their geometry and small thickness. These characteristics have made necessary a great number of very small elements to represent adequately the fuse elements and their contacts with the star core. However, a great number of elements in the simulation can produce the saturation of the software or, in any case, it makes necessary a longer simulation time.

Regarding to the simulation process, the fuse element has been considered as the heat source, so the power dissipated depends on the fuse element resistance and the current flowing through it. Experience proves that the fuse temperature maintains nearly constant along the fuse body, decreasing near the end caps. According to that, the electric power consumed by the fuse has been supposed to be uniformly distributed along the fuse length. On the other hand, the heat generated by the current flowing through the fuse elements has been considered to be transmitted by conduction inside the fuse, whereas a combination of radiation and natural convection with the external air through the surface of the ceramic tube has been supposed.

On this way, in the development of the fuse model, in addition to geometric dimensions of different parts of the power fuse (star-core, fuse elements and ceramic tube), also external conditions (ambient temperature and heat transfer coefficient) and thermal conductivity characteristics of different materials (copper, quartz sand and ceramic materials of the star-core and the fuse body) used in the power fuse have to be included.

Next sections describe two different models and compare the results of simulations with those derived from testing real fuses. The main purpose of this comparison has been to check the validity of the simplifications introduced to get a model easily applicable to any fuse without loss of accuracy. For that reason, only a portion of the fuse, corresponding to a complete loop of the fuse elements around the star-core, has been considered in a preliminary analysis.

Model A: Flat model

In this model, each fuse element has been modelled, as in the real fuse, using flat pieces disposed in a helical way around the star-core. This implied the creation of 12 pieces: 6 long pieces and other 6 smaller pieces to model the contact of the fuse elements with the star-core (Fig. 2). In this particular case, there were 3 fuse elements; so 36 pieces were necessary to model them. However, fuse elements notches have not been taken into account because the analysis is limited to current values equal or below the rated current, and the effect of the higher current density at these sections of the fuse elements is slighter than for higher currents.

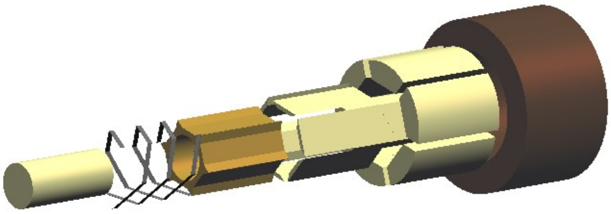

Fig. 2. Model with flat fuse elements

Model B: Helical model

The final model had to comply with the requirement of being easily applicable to any fuse, so a new model has been developed, with less parts than Model A. In this new model each fuse element has been modelled as a cylindrical helix, wounded around the star-core (Fig 3). In this model, the number of parts to model the sand between the fuse elements and the porcelain tube has been also reduced from 6 to 1, being modelled in Model B as a tube with three helical cut-outs in its inner surface.

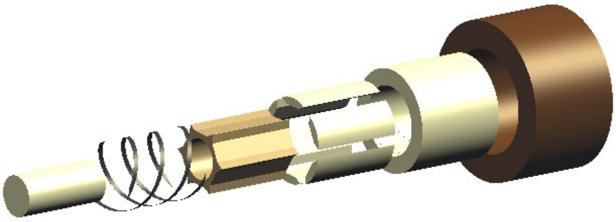

Fig. 3. Model with helical fuse elements

Comparison between models A and B

Once the models were completed, they were meshed and simulated using the Cosmos DesignStar package. With regard to the meshing process, the same options were applied to both models: linear tetrahedral elements of 1.4 mm for the whole model except for the fuse elements and the faces of the star-core in contact with them, where mesh controls were applied to limit the element size to 0.3 mm. These meshing options led to a mesh with 365244 elements and 65052 nodes for Model A, and with 396368 elements and 72050 nodes for Model B. According to this, both models are similar as for their meshing characteristics. Fig. 4 shows a detail of the meshing of both models.

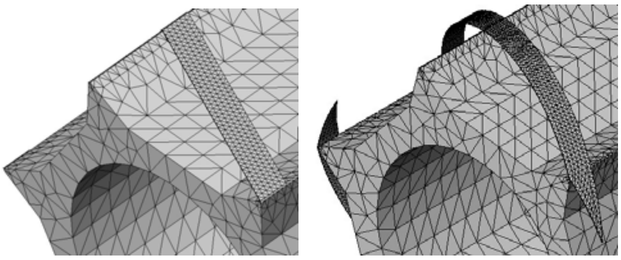

Fig. 4. Meshing of Models A and B

The conditions applied to perform the simulation were also the same for both models: radiation and natural convection by means of a combined heat transfer coefficient, and the same dissipated power than the real fuse, considering the total power linearly distributed along the fuse body. Fig. 5 and 6 show the distribution of temperatures provided by the Cosmos software tool, considering an ambient temperature of 40 ºC and the rated current.

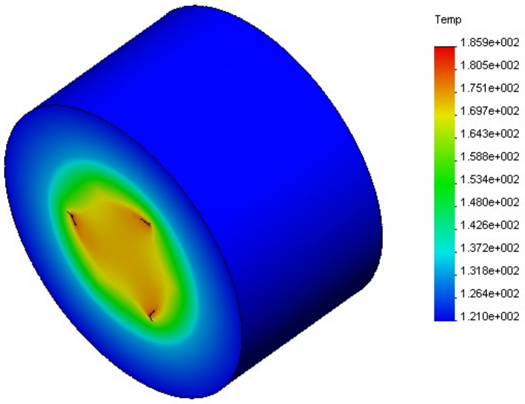

Fig. 5. Temperature distribution in Model A

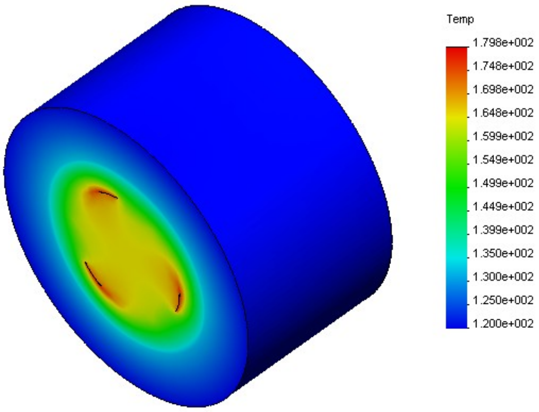

Fig. 6. Temperature distribution in Model B

The average temperatures obtained for the porcelain tube and the fuse elements are shown in Table I. These results are compared with those measured by three thermocouples placed at the middle cross-section of the fuse, on the internal and external surfaces of the tube, and on the face of the star-core in contact with the fuse elements, as specified in Fig. 7. As it can be observed, the results provided by both models are in good agreement with test results.

	Test (ºC)	Model A (ºC)	Model B (ºC)
T1	171.17	184.67	177.69
T2	133.41	133.45	132.39
T3	121.38	121.19	120.27

Table I. Comparison of simulation with real tests

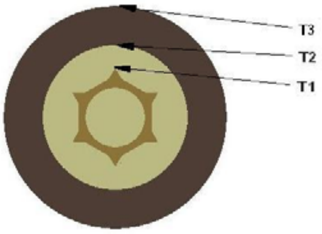

Fig. 7. Measurement points

The analysis confirms the adequacy of both models to obtain the thermal behaviour of power fuses. However, the smaller number of parts of Model B makes it more easily adaptable to derive the model of any fuse. Therefore, Model B has been finally selected to model the whole fuse.

SIMULATION RESULTS

In this section the results obtained for a complete fuse in horizontal position are presented. The model has been developed by assembling as many models B as necessary, as well as the models of the fuse terminals (Fig. 8).

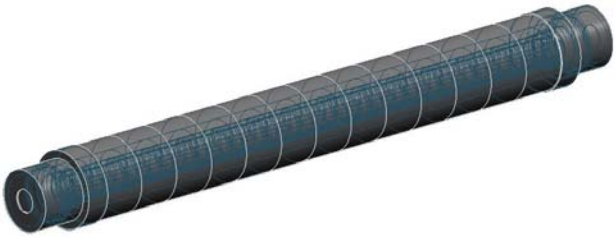

Fig. 8. Fuse model

As before, a global heat transfer coefficient has been considered, and a uniform distribution of the total power among the fuse elements, since no power is assigned to the end caps due to their smaller resistance. Fig. 9 shows the temperature distribution obtained with Cosmos, for an ambient temperature of 40 ºC and the rated current.

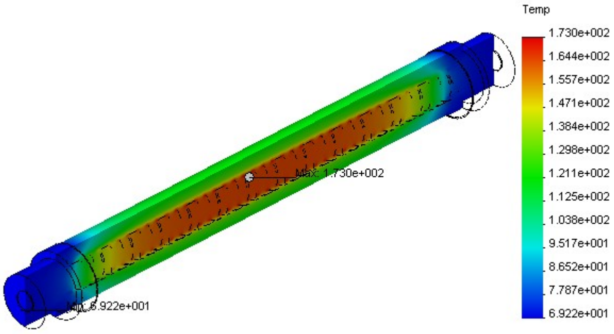

Fig. 9. Temperature distribution in the final model

Table II shows the results obtained in the simulation at the middle section as well as at the end cap, for a variety of test conditions. As it can be observed, there is a very good agreement with test results.

% I_{rated}	T_{amb} (ºC)	Test (ºC)				Cosmos (ºC)			
		T1	**T2**	**T3**	**End cap**	**T1**	**T2**	**T3**	**End cap**
100%	40	171.17	133.41	121.38	66.77	175.5	132.32	120.2	69.70
	55	190.47	149.39	135.90	80.10	191.26	147.83	134.73	83.09
	70	204.79	162.78	149.82	95.08	207.25	162.25	148.68	97.30
75%	40	110.77	90.48	84.32	53.71	109.08	89.52	83.62	56.39
	55	124.42	104.18	97.96	67.92	123.38	103.34	97.31	70.39
	70	141.31	119.46	113.15	82.84	141.57	119.24	112.51	84.99
50%	40	70.20	61.78	59.39	45.62	69.07	61.39	59.07	47.60
	55	84.46	75.96	73.60	60.05	83.74	75.72	73.30	61.85
	70	100.28	91.25	89.06	75.47	100.14	91.43	88.81	77.26

Table II. Comparison of simulation with real tests

The model proposed has been validated by means of the application of this approach to obtain the thermal behaviour of other fuses, of different sizes and rated currents. Similar results have been achieved in the analysis of those additional fuses. For example, the superficial temperature at the middle cross-section and the end cap temperature, obtained for other fuse, are included in Table III, as well as the corresponding experimental values.

% I_{rated}	T_{amb} (ºC)	Test (ºC)		Cosmos (ºC)	
		T3	**End cap**	**T3**	**End cap**
100%	40	174.88	81.22	170.09	84.67
	55	189.11	95.08	184.4	97.73
	70	212.44	113	207.56	114.73
75%	40	112.15	62.34	109.41	64.95
	55	125.17	76.47	122.53	78.41
	70	140.33	91.33	137.81	92.79
50%	40	70.90	49.75	69.68	51.28
	55	84.91	64.08	83.73	65.44
	70	99.70	79.4	98.61	80.53

Table III. Results obtained for a different fuse

CONCLUSIONS

In this paper a new model for thermal analysis of medium voltage fuses, by means of the finite element method has been presented. The model proposed is easily adapted to any fuse and has been designed and simulated with the Cosmos DesignStar package.

Temperatures obtained in simulations have been shown, considering a wide range of operation conditions, consisting in different values of current (100%, 75% and 50% of the rated current) and ambient temperature (40, 55 and 70 ºC). The comparison of simulation results with those obtained by means of real tests has allowed to verify the validity of the model developed.

REFERENCES

[1] IEC 60282-1:2005. High-voltage fuses. Part 1: Current-limiting fuses.

[2] M. Sasu, 1995, "Simulation of the heat-transfer phenomena in electrical fuses by means of the digital computer", ETEP. vol. 5, No. 3, 187-191.

[3] M.S. Agarwal, A.D. Stokes, P. Kovitya, 1987, "Pre-arcing behaviour of open fuse wire", J. Phys. D: Appl. Phys. vol. 20, 1237-1242.

[4] J. Lijun, M. Zhiying, W. Jimei, 1999, "The research of the stable temperature-rise of high-voltage current-limiting fuse-link", *Proceedings International Conference on Electrical Fuses and Their Applications*, 55-58.

[5] J.G.J. Sloot, 1987, "Analog simulations of the heat flow in a high voltage fuse", *Proceedings International Conference on Electrical Fuses and Their Applications*, 7-11.

[6] G. Hoffmann, U. Kaltenborn, 2003, "Thermal modeling of high voltage H.R.C fuses and simulation of tripping characteristic ", *Proceedings International Conference on Electrical Fuses and Their Applications*, 174-180.

[7] H. Kürschner, A. Ehrhardt, G. Nutsch, I. Harrison, A. Boerner, T. Mickley, 1995, "Calculation of pre-arcing times using the finite element method", *Proceedings International Conference on Electrical Fuses and Their Applications*, 156-161.

[8] V. Giurgiu, G. Oarga, S. Purdel, 1999, "Analysis of thermal phenomena in high-voltage fuse-links", *Proceedings International Conference on Electrical Fuses and Their Applications*, 65-68.

[9] Y. Kawase, T. Miyatake, S. Ito, 2000, "Heat analysis of a fuse for semiconductor devices protection using 3-D finite element method", IEEE Trans. on Magn. vol. 36, No. 4, 1377-1380.

[10] E.Torres, E. Fernández, A.J. Mazón, I. Zamora, J.C. Pérez, 2005, "Thermal analysis of medium voltage fuses using the finite element method", *Proceedings PowerTech*.