ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/12314005.1166029

FEM TEMPERATURE EVALUATION OF MODEL BRAKING TEST IN COMPARISON TO SELECTED LABORATORY TEST

Zbigniew Skorupka

Institute of Aviation
Krakowska Avenue 110/114, 02-256 Warsaw, Poland
tel.: +48 22 8460011 ext. 657, fax: +48 94 3426753
e-mail: zbigniew.skorupka@ilot.edu.pl

Abstract

Friction pair tests or model brake tests are one of the trials done in order to evaluate friction pair materials usefulness in full-scale brakes. Such tests are both time consuming and expensive, so most of the manufacturers want to reduce them as much as possible by using numerical simulations for both time and costs savings. It is not possible to eliminate laboratory tests entirely because of the safety regulations requirements. Without regulatory compliance, no brake material can be used in consumer products such as cars or airplanes.

Nowadays most of the FEM programs are capable of simulate braking (friction) process in many of its aspects. Accuracy of calculations varies according to complexity of the model and phenomena taken into account. One of the interesting aspects of braking is the friction temperature generation especially due to its destructive impact on the vehicle brakes. Laboratory evaluation of the temperature can be performed in limited spectrum because of inaccessibility of the friction area. FEM calculations can help to determine temperature changes and temperature value during process. It also helps to determine temperature induced damaging conditions for the friction material or the whole brake design.

In this paper author compares FEM of friction temperature calculations and laboratory test performed in Landing Gear Laboratory of Institute of Aviation in Warsaw where author works on daily basis. FEM simulation was created in order to resemble mentioned friction material test and to create model of the geometry of the tested material including test stand parts. All of the calculations were performed using COMSOL MULTIPHISICS software.

Keywords: *transport, air transport, simulation, friction materials, model braking*

1. Introduction

FEM analysis is nowadays the most common tool for estimating behaviour of many design aspects. It most commonly used for stress strain analysis in mechanics. In aviation, FEM is widely used also for aerodynamics especially for flow calculations. In brakes, FEM analysis is quite popular in temperature generation [4], sound (squeak) generation [6] and thermoelastic phenomena [5].

FEM based temperature brake calculations are focused on calculating maximal temperature during one time or repeatable braking [8] in order to determine if friction material withstands use in designed brake system. Complex approach to the brake process is still not very popular due to simplified mathematical descriptions of the friction process itself and due to high result dependence to local material behaviour. Nowadays most often FEM simulations are made with direct link to the previously made laboratory tests what gives the best results in terms of accuracy of the simulations themselves. Opposite approach is also made but it has to be proven by the corresponding laboratory tests. Thanks to FEM simulations it is possible to make first assumptions on brake behaviour in order to reduce design errors and number of (expensive in most of the cases) laboratory tests before complete brake is made.

In this article author wants to show the correlation of the one test results to the simple FEM simulation based on custom made sample braking (friction material test) model not only in terms of determining one point on temperature curve but also to calculate full temperature curve corresponding to the one obtained in the laboratory test.

Most popular software used in such calculations is based on ANSYS [7] or LSDYNA solvers (what is now one program suite called WOKRBENCH). Both mentioned programs are known to be best in their classes what comes with very good solving quality in opposition to user friendliness. Author decided to go in another direction and to use more user-friendly COMSOL Multiphisics software gaining much of the popularity due to use simplicity and quality of results.

As it was described earlier as base for the simulation one the friction material test was taken. Results from the test were then transferred to the software and temperature curve was calculated. Description of taken steps and results are described in below chapters of this article.

2. Laboratory test as base for FEM simulation

Test to be modelled in FEM software (and from which results are taken as source of date to the simulation) was carried out using IL-68 (Fig. 1) test stand. IL-68 is the inertial type test stand designed for friction pair materials testing mainly for brake use. The specificity of the IL-68 tests lies in fact that one tests is equal to one full braking from initial velocity $V_{\max}$ up to full stop ($V_{\mathrm{end}} = 0$). During typical tests [1], there is made 5 to 8 bed-ins for proper friction pair in contact zone geometry cohesion (in order to obtain maximal use of friction area). Next, 10 qualification test are carried out in order to evaluate various properties of braking process such as friction coefficient, wear, temperature or braking time in designated conditions.

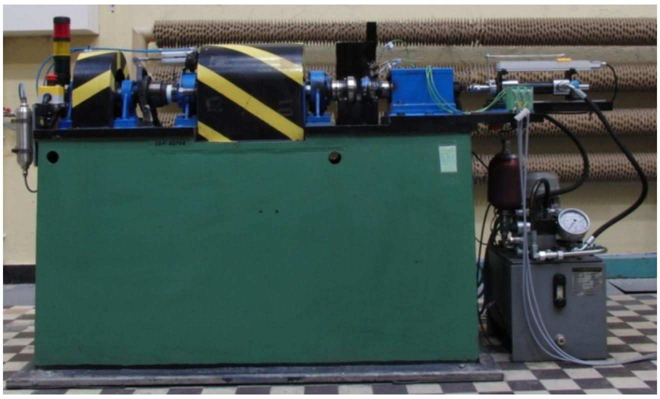

Fig. 1. IL-68 test stand side view

Test object in model tests was a set of samples (sectors) made out of real friction pair materials (Fig. 2).

Fig. 2. Example of model friction pair used in IL-68 tests (right friction material samples, left cast iron counter sample)

IL-68 test stand allows to measure and to acquire number of test parameters such as braking torque, braking force, braking time, and rotation speed of samples. These parameters are used not only for measurement purposes but also as internal control signals for IL-68 test stand. Temperature measurement is made by external system. In test that is the base for this article, temperature was measured by thermocouple [2] placed on the cylindrical surface of friction material sample near shaft of the test head. All of measurements are acquired to PC computer for further analysis.

3. FEM software used in simulation

For all simulations, COMSOL Multiphisics software was used. COMSOL Multiphisics is a software platform in which most important part is simulation package solving nonlinear partial differential equations using finite element analysis in one, two and three dimensions. Program architecture allows simulating direct interactions between different types of physical interactions in order to obtain full view of simulated phenomenon/phenomena. Software is constructed in a manner that makes possible to simulate coupled phenomena within one interface where calculated results are transferred from one module to another using automatic or manual mode. Most of the definitions are based on specific mathematical description (i.e. using common equations) instead of program specific one. FEM model generation can be done automatic where all of the parameters are controlled by the modelled phenomena. This approach makes much easier (sometimes even possible) to create useful FEM simulation by the people not being specialists in using that kind of software. Downside of described approach is possibility in achieving less accurate results than using much more specialised, complex and less user-friendly software. COMSOL Multiphisics is able to perform complex computations and modelling using broad variety of techniques (ex. direct programming, precise mesh elements definitions, defining users' differential equations etc.) what still makes it very powerful tool. It is necessary to say that even during use of the COMSOL software it is crucial to know at least basics of simulated phenomenon or phenomena to avoid situation where results of simulations are far from reality without noticing it – what is a common mistake to make. Both modelling and simulation described in this article were made using COMSOL Multiphisics 5 software.

4. Geometrical model for FEM analysis

Geometrical model created for the FEM use is a simplified resemblance of real life test samples used in friction material tests performed using IL-68 stand. Simplifications are the results of limitations of FEM program and need for computation power what results in optimized calculation speed. Model simplification was intended not to affect overall result accuracy as well as not allow obtaining too high error value. Geometric model was created as 3D solid.

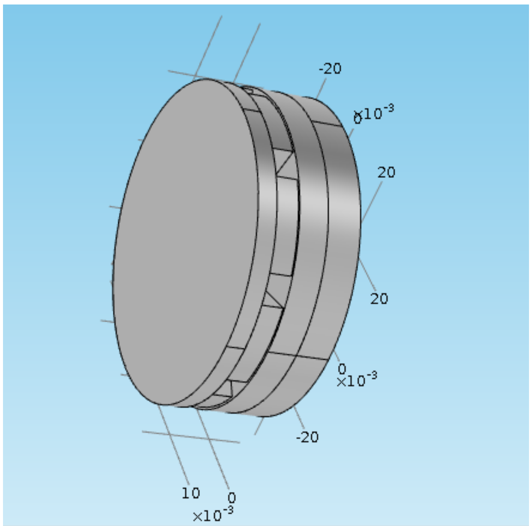

Fig. 3. Geometrical model for FEM analysis

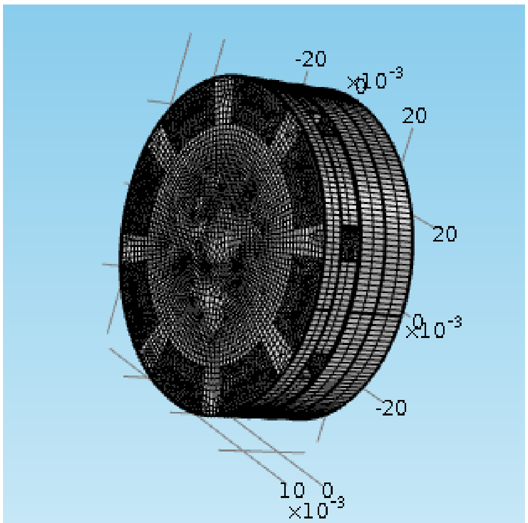

Fig. 4. Meshed FEM model

Created model consists of friction pair components (non-rotating sample and rotating counter sample) and tests heads components that are in direct contact with friction pair. Modelled test heads components have great influence on heat transfer so their omission will result in serious falsification and distortion of the computed results.

Mesh was generated with use of automatic program settings according to simulated phenomenon. Such meshing process results (as software manufacturer states) in well-optimized mesh. Generated mesh contains triangular elements in friction area and quadrilateral elements in the rest of the model. Whole model is constructed out of the 56814 elements total.

5. Mathematical model used in FEM analysis and simulation parameters

As it was stated earlier, theoretical model was generated to resemble heat generation/dissipation from friction material test. FEM method used base on several equations described in this chapter.

Heat transfer in solid equation:

$$\rho C_p \frac{\delta T}{\delta t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (k \nabla T) + Q, \tag{1}$$

where:
ρ – density,
C_p – specific heat,
k – heat conductivity,
T – temperature,
Q – heat sources,
$\mathbf{u}$ – velocity.

The joint conductance between friction pair surfaces is described by the formula:

$$h = h_c + h_g + h_r, \tag{2}$$

according to assumption that conductance h is a sum of three conductances: the constriction conductance, h_c, from the contact spots, the gap conductance, h_g, due to the fluid at the interstitial space and the radiative conductance, h_r.

The Cooper-MikicYovanovich (CMY) [9] correlation is valid for isotropic rough surfaces and assumes plastic deformation of the surface asperities. It relates h_c to the asperities and pressure load at the contact interface:

$$h_c = 1.25\, k_{contact} \frac{m_{asp}}{\sigma_{asp}} \left(\frac{p}{H_c} \right)^{0.95}, \tag{3}$$

where:
H_c – microhardness of the softer material,
p – contact pressure,
m_{asp} – average slope of microscopic surface asperities,
σ_{asp} – average height of microscopic surface asperities,
$k_{contact}$ – harmonic mean of the contacting surface conductivities.

The gap conductance h_g (assuming that the interstitial fluid is a gas) is defined as:

$$h_g = \frac{k_g}{Y + M_g}, \tag{4}$$

where:
k_g – gas conductivity,
Y – mean separation thickness,
M_g – set of gas parameters.

In this simulation there was $h_g = 0$ assumption used, according to the program manual suggestion.

Radiative conductance h_r needs to be considered at temperatures above 600°C and is described by following formula:

$$h_r = \sigma \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d}(T_u^3 + T_u^2 T_d + T_u T_d^2 + T_d^3)\,. \tag{5}$$

The friction heat, Q_{fric}, is partitioned in to $r\,Q_{fric}$ and $(1 - r)\,Q_{fric}$ at the contact interface. If the two bodies are identical, r and $(1 - r)$ would be 0.5 so that half of the friction heat goes to each surface. However, in the general case where the two bodies are made of different materials, the partition rate might not be 0.5. The Charron's relation defines r as

$$h = \frac{1}{1+\xi_d} \quad \text{and} \quad \xi_d = \sqrt{\frac{\rho_u\, C_{p,u}\, k_u}{\rho_d\, C_{p,d}\, k_d}}\,, \tag{6, 7}$$

and accordingly, $(1 - r)$ is:

$$(1-h) = \frac{1}{1+\xi_u} \quad \text{and} \quad \xi_u = \sqrt{\frac{\rho_d\, C_{p,d}\, k_d}{\rho_u\, C_{p,u}\, k_u}}\,, \tag{8, 9}$$

Simulation was created out of parameters, which were taken from real time test. Summary of the data set is shown in the Tab. 1. It is necessary to know that all of the used values were directly connected with real time test. Parameters shown in designation column as time dependent (ex. $n(t)$) were described in the simulation as functions changeable in the time of the process. Functions were determined as tabularised data sets taken directly from measurements.

Tab. 1. Main FEM simulation parameters used

N°	Name	Designation	Unit	Source
1	Moment of inertia	I	kgm^2	test parameter
2	Ambient temperature	T_pow	K	test parameter
3	Average friction radius	r_pc	m	test parameter
4	Braking time	t	s	recorded during test
5	Rotational speed	n(t)	rpm	recorded during test
6	Braking torque	Mh(t)	daNcm	recorded during test
7	Angular speed	omega(t)	rad/s	computed from recorded measurements
8	Friction coefficient	us(t)	–	

6. Obtained results and discussion

Simulation solution is the temperature curve versus time. Model and its solution is 3D what gave the possibility to evaluate temperature in whole model volume (Fig. 4.). For this article purpose, temperature was determined for one temperature measurement point (T_3 thermocouple)

In Fig. 5, there is the comparison between temperature T_3 obtained in the tests made on IL-68 test stand and calculated using FEM analysis. It can be directly seen both curves are converged enough in its trend as well as in value of end point temperature. Rest is not so near especially that measured temperature is visibly influenced by other phenomena while calculated one does not show this. Calculated curve looks like being interpolated (ideal one) without any fluctuations seen on the real one. Ideal curve is most probably calculated because of omitting some of the real phenomena in the braking process. According to the assumption that calculations are simplified comparing to the real process it can be said that curves match themselves with acceptable accuracy [3].

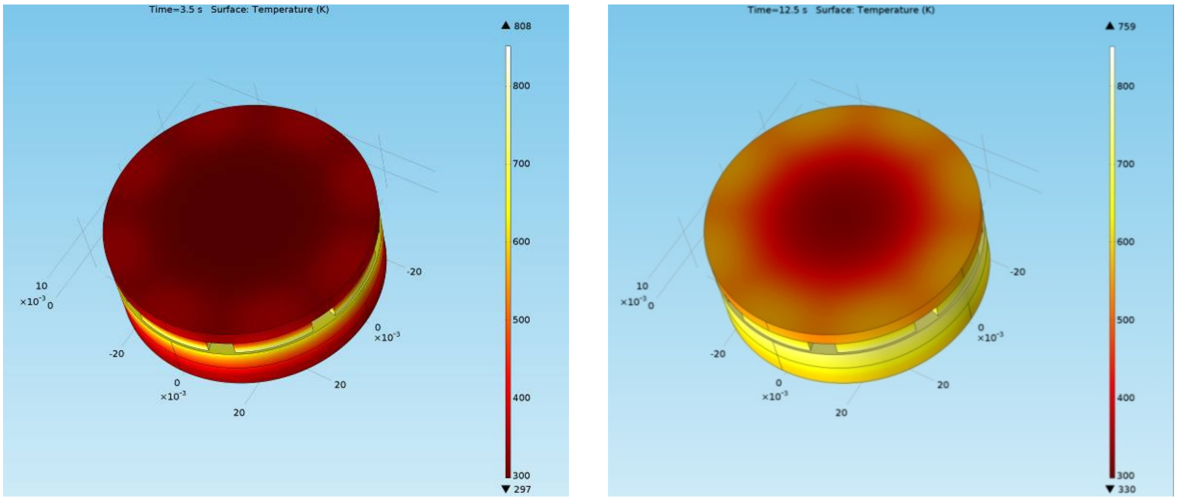

Fig. 4. Sample frames of FEM model temperature calculations

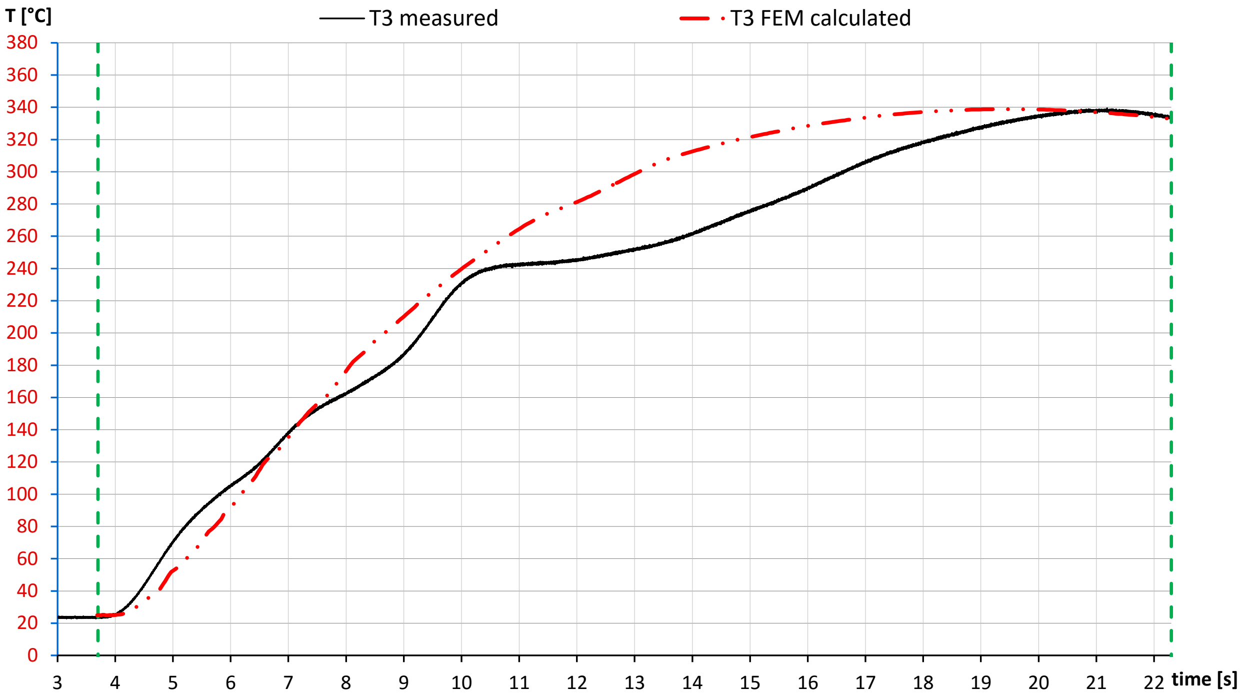

Fig. 5. Comparison between curves of measured and FEM calculated temperatures

Discrepancy between two curves has its source in two areas. First is the geometrical model simplifications and the second is the mathematical description of analysed process.

Geometrical model simplification is a result of compromise between real life test object and software/hardware limitations. Nowadays hardware (i.e. computation power) is not an issue in terms of computation itself and can be only taken into account is time of the process. More complex model is more time is needed to obtain results. Software limitation however results in fixed approach to model design. Most of the FEM programs are more or less sensitive to the steps of the model construction itself. Scale of simplification must be limited enough in order to be as close as possible to the real life object. Most of the FEM programs do not like any small rounds, fillets or other edge treatments; also, there is the problem with continuity of the surfaces. In this particular model was necessary to remove some of the grooves on the top of the friction pair.

Also meshing process itself is source of errors in computations. In order to obtain best results mesh elements should be as small as possible and also as dense as possible. Results (in theory) would be the best with such approach but in reality too dense mesh can be not computable due to software limitations or so time consuming that ratio between result accuracy and time needed is not justified. Automatic physics oriented mesh generation process is quite functional an accurate but it also optimized in order to obtain acceptable computation speed what result in less accurate results.

Last and most important source of errors is mathematical method used in order to evaluate temperature. It bases on basic dependences in coupled friction temperature determination (by coupled it is meant to calculate heat flux due to friction and later its distribution in model volume). Equations used for heat generation description (explained in previous chapter) are simplified in principle what mostly affect on results. It is necessary to remember that braking process itself is so complicated that yet there is no mathematical model that can describe it fully.

7. Summary

In FEM, analysis laboratory test results were used as input parameters. Simulation was constructed in the manner to resemble as much as possible (within simplifications) real life friction sample test. Simulation was designed to be coupled between friction heat generation and heart flux penetration in friction material volume using heat transfer in solids module.

– FEM geometrical model was built in with some simplifications needed in order to maintain efficiency and optimisation of the computation. Simplification did not influence models' integrity and mapping compared to real one,
– Calculation model used in FEM software, enabled to get results mainly consistent with measured ones. Occurred errors were at the level of 20% for maximal T_3 value and 8.8% for average T_3 value. Divergence (in terms of curve shape) between measured temperature and calculated one can be explained by use of mathematical model that is simplified, and does not cover all of the phenomena during braking. Obtained result is sufficient from the engineering point of view but should be much more converged for scientific analysis.

References

[1] Skorupka, Z., Grygorcewicz, P., *Badania laboratoryjne hamulców ciernych w laboratorium badań podwozi lotniczych*, Technika Transportu Szynowego, 2013.

[2] Skorupka, Z., *Pomiar temperatury pary ciernej w badaniach modelowych*, Logistyka, 3/2015.

[3] Baranowski, P., Damaziak, K., Jachimowicz, J., Małachowski, J., *Symulacja stanów nieustalonych procesu hamowania w hamulcach pojazdów kołowych*, Logistyka, 3/2011.

[4] Kułakowska, A., Patryk, R., *Analiza numeryczna procesu nagrzewania się tarczy hamulcowej podczas hamowania*, Logistyka, 6/2011.

[5] Aramburu, E., Calvo, R., *Brake cooling simulaton: A combined procedure of CFD thermal and 1D software*, Proceedings of the FISITA 2012 World Automotive Congress, 2012.

[6] Liu, X., Wang, H., Shan, Y., He, T., *Nonlinear transient dynamic analysis of disc brake squeal using improved Hilbert-Huang transform*, Proceedings of FISITA 2012 World Automotive Congress, 2012.

[7] Galindo-Lopez, C. H., Tirović, M., *Air flow and heat dissipation analysis of commercial vehicle brake discs*, 11th EAEC European Automotive Congress, 2007.

[8] Meachair, D. O., Elliot, G., Jelić, S., Fechner B., Bhambare, K., *Brake Duty cycle simulation for thermal design of vehicle braking system*, Proceedings of the Eurobrake 2013 Conference, Dresden 2013.

[9] *Heat Transfer Module User's Guide*, Comsol Multiphisics AB, 2014.