JAN PERNER TRANSPORT FACULTY
UNIVERSITY OF PARDUBICE

Proceedings of (sborník příspěvků)

THIRD SCIENTIFIC CONFERENCE
TŘETÍ VĚDECKÉ KONFERENCE

Part I

Sections:

1 Transport means
2 Transport infrastructure
6 Electrotechnics in transport and communications
7 Operational reliability of transport means and infrastructure

Díl I.

Sekce:

1. Dopravní prostředky
2. Dopravní infrastruktura
6. Elektrotechnika v dopravě a spojích
7. Provozní spolehlivost dopravních prostředků a infrastruktury

Pardubice, 2003

Table of Contents (obsah):

Section 1

Transport means (dopravní prostředky)

Jan Perner Transport Faculty
UNIVERSITY OF PARDUBICE
THIRD SCIENTIFIC CONFERENCE OF JAN PERNER TRANSPORT FACULTY
„New trends in transport and communications“,
Pardubice 11 – 12 September 2003

INFLUENCE OF THE FE DISCRETIZATION ON ACCURACY OF CALCULATION OF CONTACT STRESS IN A SYSTEM WHEEL - RAIL

Prof. Aleksander SŁADKOWSKI, Tomasz KUMINEK M.Sc., Silesian Technical University, Krasińskiego str. 8, 40-019, Katowice, Poland, sladk@polsl.katowice.pl

1. Introduction

The majority of research of contact interacting in the pair of wheel - rail has been carried out with usage of the approach of Hertz. Earlier authors designed the analogous approach basing on the Hertz theory allowed to consider contact interacting of wheels and rails at presence of several contact zones, and also nonzero angles of attack [1]. The software engineering, basing on FEM application, has allowed to use the given method for the solution of contact problems. Usage of a finite element method has allowed to eliminate the errors of evaluations connected to assumptions of the Hertz theory.

In programs allowing to analyze contact problems with the help of FEM accuracy of calculations is influenced by many parameters among which the main are density of a FE mesh and relative arrangement of nodes on contacting surfaces. Comparison of the contact stress determined according to Hertz theory and with the help of FE calculation using program MSC.MARC is submitted.

2. Analytical calculation

The problem of interaction of elastic cylinders according to the Hertz theory has been selected as base test model for inspection of a design procedure with usage of the FEM. Thus interaction of two elastic semicylinders of equal radius (r = 0,5 m) which nestled to each other force (P =1,0e+8 N) was considered. As a material of viewed bodies ideal steel with the following characteristics was used: a coefficient of elasticity E = 2,0e+11 MPa, a Poisson's ratio ν=0,32. It is supposed also according to the Hertz theory that between interacting bodies friction misses.

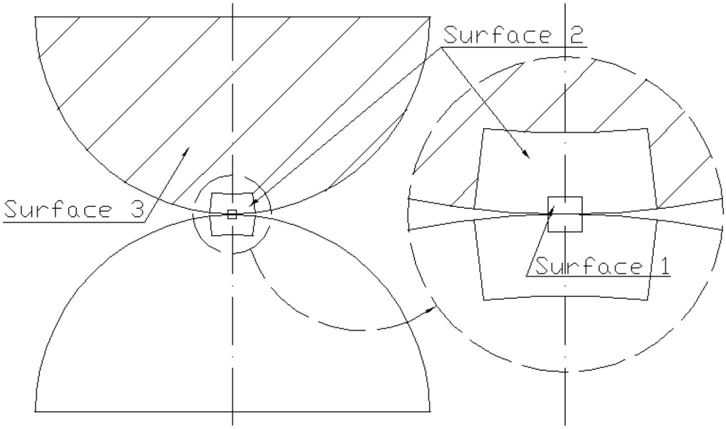
According to the book [2] maximal contact stresses determined by means of the Hertz theory can be calculated with usage of the following formula

$$\sigma_{\max} = 0{,}5642\sqrt{\frac{PE}{lr\ (1-\nu^2)}} \qquad (1)$$

Here too it is accepted, that the length of considered cylinders is equal l = 10 m. After substitution of calculation data it is obtained that σ_{max} = 1,191e+9 Pa. The obtained maximum stress was used further for comparison with results of numerical calculations with usage of the FEM.

3. Analysis of the test model

The program MSC.Visual NASTRAN for Windows 2002 has been used for making КЭ of model of contact interacting elastic semicylinders. For the creation of model properties of finite elements were determined based on usage of an ideal - elastic material and plain strain. Generation of a finite - element mesh was carried out in an automanual regime. I.e. first the geometry of model which is shown on pic. 1 was set.



Pic.1 *Geometrical model of the investigated test problem*

According to an above mentioned figure each of semicylinders is divided on three zones (surfaces). Thus a size of a surface 1 was selected according to width of a contact zone which was determined with usage of the known formula

$$b = 1{,}522\sqrt{\frac{rP}{2El}} \tag{2}$$

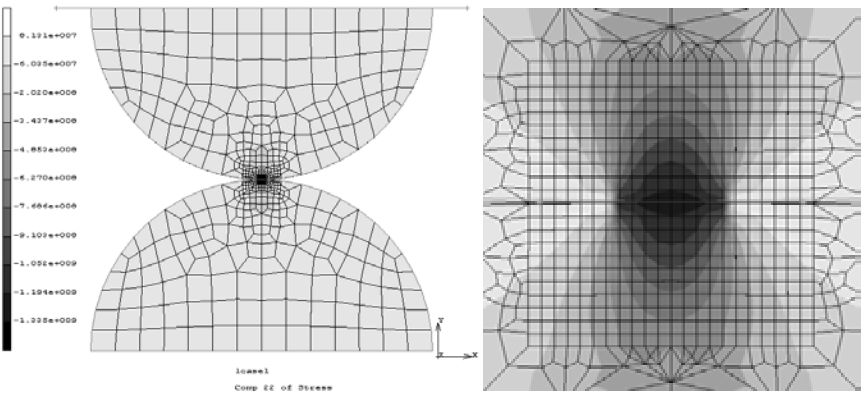
where b - a halfwidth of a contact zone according to the Hertz theory. It was accepted, that the width of a zone 1 is equal 4b, and the height - 2b. Thus the constant FE discretization was manually set on borders of a surface 3 and then the constant FE mesh for a surface was automatically generated.

The definition of an amount of elements and their allocations to each of boundaries was carried out by hand-carry. Thus allocation of elements to the boundary of a surface 3 was primary. The amount of elements along curves of surface 1 was variable. Therefore an amount of elements in a area 1 and on a contact surface was both variable. The amount of elements on boundaries of area 1 was selected such that the created mesh was the regular and consisting only from tetragonal elements. The specified mesh is selected because the mesh consisting of only triangular elements gives a major error of calculation. Generation of a FE mesh for area 2 was dependent on grids for areas 1 and 3. Such sampling of the FE generation has been stipulated by necessity of a modification of density of a mesh for near contact zone without an essential modification of a mesh in a prime area of cylinders.

Conjugate nodes formed on boundary curves then "were sewed together". The finite - element mesh for the lower semicylinder was formed analogously. Thus meshes of surfaces 3 upper and lower semicylinders were identical, and meshes of surfaces 1 could different.

The FE model previously created by the program MSC.Visual NASTRAN further is exported to the program MSC.MARC 2003 where boundary conditions are set. Thus nodes were on a diametrical straight line of bottom semicylinder are set by the fixed, and nodes on a diametrical straight line of top semicylinder can have an equal vertical displacement. The specified displacement is not set and determined as result of action of total force preset on the given boundary. Parameters of a solved problem, such as types of used elements or the size of operative memory assigned for a task are in addition set.

As a result of the carried out calculations distribution of stresses in analyzed semicylinders are determined. On pic. 2 the example of a stress σ_{yy} distribution is shown for the FE model containing of 200 elements in near contact zone (surface 1). On pic. 2b the contact zone of the cylinder is shown in the increased scale.



a b

Pic.2 Distribution of stress σ_{yy} for a test problem containing 200 elements in near contact zone (surface 1)

Comparison of numerical results of calculation for FE models with various density of meshes with analytical calculation is instanced in the table 1. Apparently from the presented comparison accuracy of the solution in basic depends not so on density of a mesh, however from amount of node pairs which are formed in a contact zone and their distribution.

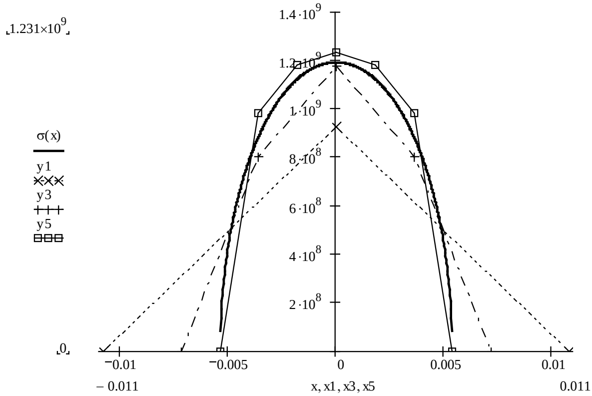
If to compare the received numerical results to the solution received according to Hertz theory it is necessary to determine criterion of an estimation of such decisions. For example, if to compare the maximal normal stress σ_{yy} only for the first numerical experiment when one contact pair nodes is formed, the relative error exceeds 10%. Whether such criterion is correct? Moreover, such criterion does not explain the certain ambiguities. For example, when it is formed only 3 pairs of contact nodes (numerical experiment nr 3) the error by the given criterion is equal of 1,4%. While for numerical experiment nr 9 at presence of 7 contact pairs of nodes the error makes 4,9%.

The explanation to this can be found if to draw schedules of the contact stress determined by means of Hertz theory and numerically with use of the FEM. On fig. 3 thick continuous line represents distribution of contact stress determined by means of Hertz theory according to the formula

$$\sigma_{yy}(x) = \sigma_{\max} \frac{\sqrt{b^2 - x^2}}{b} \qquad (3)$$

Number of numerical experiment	Amount of elements on a surface 1	Amount of the formed contacting pairs of nodes after a loading	Distance from origin [m]	Contact node force [N]	Maximum stress σ_{yy} [Pa]	Error of calculation [%]
1	2	1	0	1e8	0,926e9	53,633
			0,0108	0	0	
2	8	3	0	6,157e7	1,14e9	42,652
			0,0054	1,921e7	3,558e8	
			0,0108	0	0	
3	18	3	0	4,228e7	1,174e9	17,786
			0,0036	2,886e7	8,017e8	
			0,0072	0	0	
4	24	5	0	3,214e7	1,20e9	14,247
			0,0027	2,735e7	1,013e9	
			0,0054	6,443e6	2,386e8	
			0,0081	0	0	
5	50	5	0	2,575e7	1,192e9	9,018
			0,00216	2,337e7	1,082e9	
			0,00432	1,376e7	6,370e8	
			0,00648	0	0	
6	72	5	0	2,215e7	1,231e9	8,082
			0,0018	2,121e7	1,178e9	
			0,0036	1,770e7	9,833e8	
			0,0054	0	0	
7	98	5	0	2,011e7	1,306e9	25,475
			0,00154	1,997e7	1,297e9	
			0,00308	2,011e7	1,306e9	
			0,00462	0	0	
8	128	7	0	1,639e7	1,214e9	5,13
			0,00135	1,588e7	1,176e9	
			0,0027	1,442e7	1,068e9	
			0,00405	1,150e7	8,519e8	
			0,0054	0	0	
9	162	7	0	1,499e7	1,249e9	17,201
			0,0012	1,468e7	1,223e9	
			0,0024	1,409e7	1,174e9	
			0,0036	1,373e7	1,144e9	
			0,0048	0	0	
10	200	9	0	1,304e7	1,207e9	3,57
			0,00108	1,277e7	1,182e9	
			0,00216	1,196e7	1,107e9	
			0,00324	1,056e7	9,778e8	
			0,00432	8,182e6	7,576e8	
			0,0054	0	0	
11	242	9	0	1,206e7	1,228e9	15,005
			0,000982	1,188e7	1,21e9	
			0,001964	1,133e7	1,154e9	
			0,002946	1,064e7	1,084e9	
			0,003928	1,011e7	1,03e9	
			0,00491	0	0	

Tab. 1 Comparison of the calculation results for different FE meshes



Pic.3 Comparison of distributions of stress in a contact zone determined by means of Hertz theory and numerically with help of FEM (Hertz theory - a continuous thick line, FEM numerical experiments nr 1 - a dotted line, nr 3 - a dot-dash line, nr 6 - a continuous thin line)

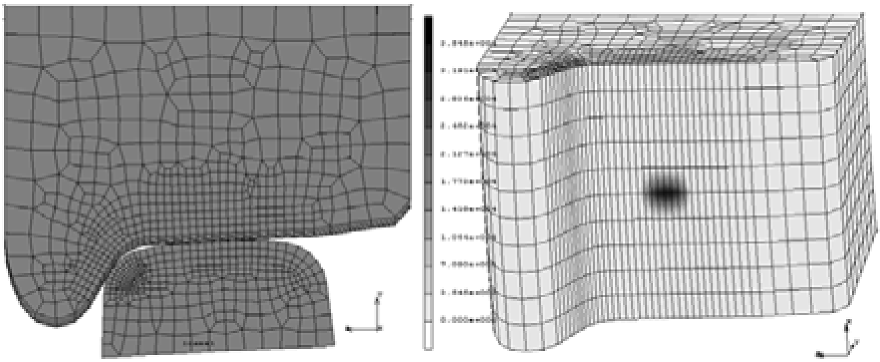
Other three schedules represent results of numerical experiments with use of the FEM. Apparently from these schedules we can speak about satisfactory accuracy of the solution only for the schedule corresponding to presence of five contact nodal pairs. Therefore for an estimation of accuracy of the received numerical solutions the following criterion has been chosen.

$$\delta = \frac{\int_{-2b}^{2b} \left[\sigma_{yy}(x) - y(x)\right] dx}{\int_{-b}^{b} \sigma_{yy}(x) dx} \times 100\% \qquad (4)$$

In the formula (4) the received numerical solutions are used as function y(x). Specified integral were determined with use of the program MathCAD. In table 1 in last column errors of calculations was defined with use of the formula (4). Analyzing the received results it is possible to make the conclusion, that accuracy of the solution practically does not depend directly on density of a FE mesh in a near contact zone. For example the FE grid in the specified zone in numerical experiment nr 11 has in comparison with experiment nr 8 in 2 times more of finite elements. Thus the error of such solution in 3 times is higher. Density of a FE mesh in a near contact zone influences on accuracy of the solution indirectly. The amount of formed contact nodal pairs depends on such density, that in turn influences on accuracy of the solution. It proves that fact (see to tab. 1) that at formation of 3 contact pairs the average error of the decision is equal 30,2%, at 5 pairs – 14,2%, at 7 pairs – 11,2%, at 9 pairs – 9,3%.

The analysis of schedules of distribution of contact stress shows that at formation in a contact zone of 7 and more contact pairs accuracy of the received solution is satisfactory. Accuracy of the solution in the big degree depends on correctness of choice of a FE grid at an arrangement of contact nodes. If it was possible to arrange nodes so there are nodes situated near the border of a contact zone then accuracy of the solution considerably grows. As an example it is possible to compare results of numerical experiments 10 and 11 where at identical amount of contact pairs accuracy of the solution changes more than in 4 times.

The carried out researches of accuracy of the solution were used for an estimation of accuracy of solutions received by means of the described techniques for research of contact in the pair wheel - rail. As an example of such calculations the problem of contact interaction of a wheel and a rail with profiles of working surfaces executed according to standards of the countries of the former USSR is shown on pic. 4.



a b

Pic.4 Example of the solution of a problem of contact interaction in the pair wheel - rail

In the resulted example interaction of a wheel and a rail is considered at the moment of formation of the second (flange) zone of contact. The following conditions of loading of a wheel have been accepted. The vertical force operating on a wheel, is equal 200 kN (dynamic effect was taken into account in quasistatic statement). The transversal force is equal 2 kN. For the specified loading and created FE meshes in two contact zones 8 contact nodal pairs are formed. At increase of transversal loading there is a change of stresses and sizes of contact zones. Thus the flange zone, and also stresses in it, is increased, and the central contact zone is on the contrary decreased. In view of that the amount of contact nodal pairs remains approximately fixed, the error of the obtained solution should not exceed 10%.

◆◆

Summary

Применение МКЭ является одним из наиболее эффективных способов решения задач контактного взаимодействия в паре колесо – рельс. Для оценки точности решения с использованием МКЭ было проведено сравнение численных экспериментов с различной плотностью КЭ сетки в приконтактной зоне с решением Герца. Определены погрешности таких решений. На основе проведенных исследований можно было судить о точности решения задачи контактного взаимодействия колеса и рельса, пример которого также приводится.

Literature

1. Yessaulov V., Kozlovsky A., Sladkovsky A., etc. Studies into Contact Interactions of Elastic Bodies for Improvement of Wheels and Rails // Contact Mechanics IV.- Southampton, Boston: WIT Press, 1999. - P.463-472.

2. Пономарев С.Д., Бидерман В.Л., Лихарев К.К., etc. Расчеты на прочность в машиностроении. – Москва: Машгиз, 1958. – Т.2. – 974 с.