ISSN 1392 - 1207. MECHANIKA. 2005. Nr.3(53)

Accuracy analysis of the solution of spatial contact problem by means of the FEM

A. Sładkowski
Silesian University of Technology, 8 Krasinskiego str, 40-019 Katowice, Poland, E-mail: Aleksander.Sladkowski@.polsl.pl

1. Introduction

The finite element method is an effective means for the solution of the contact problems. It allows considering a real geometry of interacting bodies, complex physical and mechanical properties of materials, mixed boundary conditions. There are many software packages for such calculations, such as ANSYS, NASTRAN, ALGOR, ABAQUS and others. In particular, the author for research of contact interaction of wheels and rails used package MCS.MARC [1, 2]. While using numerical methods the special attention should be given for the questions of accuracy of the problems' solution. As was it is specified in work [3] an insufficient attention to such questions as error can reach 25 % and more. One of the objects of interest of the author the contact interaction in the friction pair wheel - rail. For the given class of issues in spatial statement the questions of the solution accuracy are especially actual.

2. Test problem of contact interaction

In the work [3] specified above planar problems of smooth bodies contact were considered. If it is possible to agree with the concept of smoothness at research of contact interaction of the wheels and rails up to any degree, for example, considering wet or greasy rails, but to consider contact of a wheel - rail in planar statement is inadmissible. Therefore, it is necessary to extrapolate conclusions received in work [3] to the contact problems in a three-dimensional statement.

The calculation of a test problem also should be based on Hertz solution and in this case expediently stop on most the simple statement. The problem of contact of two areas limited by hemispheres is the following. It is obvious, that the solution should be considered in elastic area; accordingly materials of contacting bodies should be ideally elastic. Radiuses of interacting bodies have been chosen identical and equal $r = 0.5$ m. the Material must be ideally elastic ($E = 2.0 \times 10^{11}$ Pa, $\nu = 0.32$).

From the view point that it is usually necessary to solve a significant amount of contact problems with enough dense FE meshes, what demands big expenses of calculation time, it is necessary to find a way of its economy. The previous planar problem [3] was solved in the following statement: the bottom body was fixed on the bottom surface; the set force was put to the top body by means of the additional rigid body glued to the top surface.

It is obvious, that under the influence of this force the considered surface moved in parallel, the starting position being on the calculated size of elastic displacement. Such approach can certainly be used and in spatial case. But it has the drawbacks connected with greater time of the calculation. The algorithm is developed in the program MSC.MARC, which consists of the following. The problem is solved in displacements. Therefore, on the top surface the initial displacement is set, the contact problem for such displacement is solved depending on it the total compressing force definition, according it compared with set force, then the initial displacement is corrected. As the result, for finding the solution an iterative algorithm is used. But it is quite obvious, that on top surface of the top body not total force, but the displacement force can be set. Such approach is absolutely equivalent to previous, but allows avoiding long iterations and excludes the usage of additional rigid body. Thus, for nodes on the top surface it is enough to set the boundary conditions in displacements. In of the chosen coordinate system shown on Fig. 1, for i-th node which is on the top surface, the following boundary conditions are formulated

$$\begin{cases} u_i = 0 \\ v_i = \Delta \\ w_i = 0 \end{cases} \tag{1}$$

where u_i, v_i, w_i are nodal displacement in the direction of corresponding coordinate axes; Δ is set displacement (settlement) of the top surface.

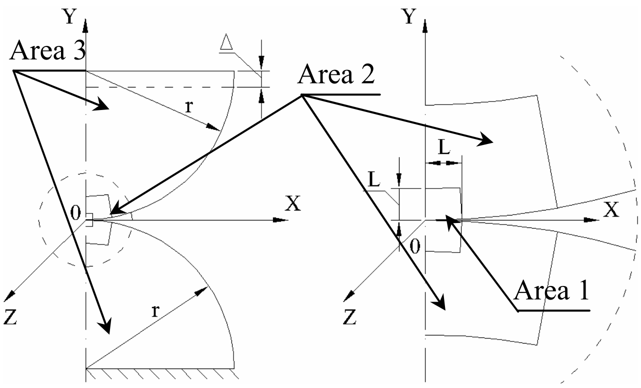

Fig. 1 Geometrical modelling of the tested three-dimensional problem

The modelling is focused on the following algorithm. The radial section of each body is broken into 3 zones. Borders digitization of the zones is carried out previously. Further in each of them the FE meshes consisting of the elements of the Plot Only type are generated. Such elements are intended only for drawing since they carry the information exclusively about geometrical properties. In zone 1 (near contact) the regular FE meshes are generated, which dimension changes depend on concrete task. The zones 3, which are more distant from the contact, have constant FE meshes. The average zones 2 are intended for the association of FE meshes of considered bodies as a whole. At FE meshes generation in zones 2 and 3 (irregular) the parameter Max Element Aspect Ratio equal to 1, 2 is used. The given parameter characterizes the ratio of the maximal geometrical size of an element to minimal one, i.e. elongation of an element. It is known, that the FE meshes made of extended finite elements give rather high calculation error.

In Fig. 2 the digitization of radial section of the top body on the FE meshes consisting of Plot Only elements is presented. Thus, depending on the density of FE mesh in a near contact zone FE meshes change only in zones 1 and 2. In particular, in Fig. 2, b the example of a FE mesh is shown when only one finite element is created in the zone 1. For comparison the case of dense enough FE mesh in the given zone is shown (Fig. 2, c) when the FE mesh consisting of 49 (7x7) elements in this zone is generated.

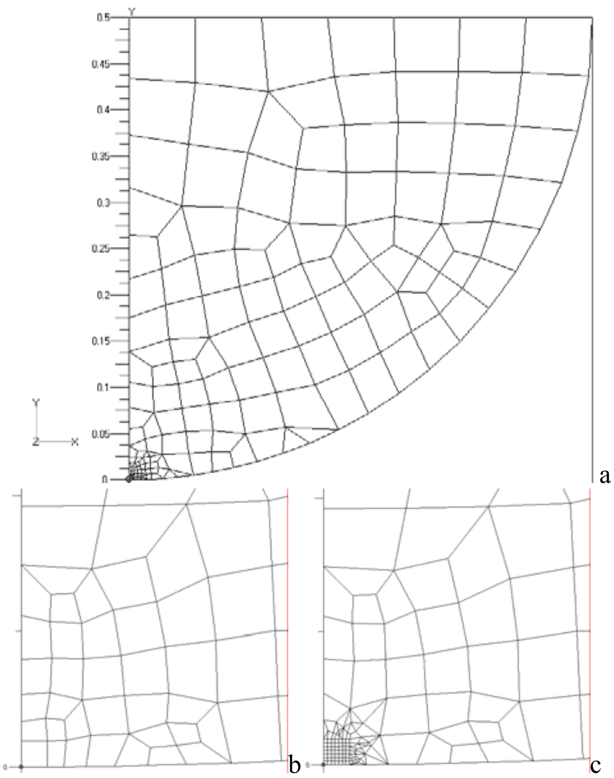

Fig. 2 FE digitization of radial section of the top body: a - general view of FE digitization with the application of Plot Only elements; b - near contact area with the digitization of a zone 1 on 1x1 element; c - near contact area with the digitization of a zone 1 on 7x7 elements

Special interest causes the size of given near contact zones. It is obvious, that it should depend on calculated radius of the contact zone. In this case at contact of the bodies limited by hemispheres, the contact ellipse turns to a circle, and its half-axes a and b [4] are equal to the radius of the given circle

$$a = b = \sqrt[3]{\frac{3rF\left(1-\nu\right)^2}{4E}} \tag{2}$$

It is obvious, that at the set geometry and properties of contacting bodies the sizes of contact zone will depend only on the size of compressing load. On the other hand, loading should be such that the contact zone was whenever possible greater, but contact stress thus should not exceed the limit of yield strength plasticity for the considered material.

The maximal contact stress is defined by the formula

$$\sigma_{max}^{H} = \frac{1}{\pi}\sqrt[3]{\frac{6FE^2}{r^2\left(1-\nu^2\right)^2}} \tag{3}$$

For the set compressing force F the approach of contacting bodies can be certain also

$$\Delta = \sqrt[3]{\frac{18F^2\left(1-\nu^2\right)^2}{rE^2}} \tag{4}$$

From the new format that calculations were carried out for various compressing forces and geometrical modelling should be uniform, as the result of the carried investigations the size of the near contact zones (Fig. 1) has been chosen: $L = 0.0024$ m. Unlike the planar problem considered before the size of near contact areas was reduced more than 4 times, and it has led to the necessity of finer digitization for the given area and, accordingly, to the increase in amount of nodes of a plane mesh, that, in turn, has led to essential increase in the number of nodes of three-dimensional FE meshes and the general number of the degrees of freedom. The size of the approach has been chosen also $\Delta = 9\times10^{-6}$ m. Thus, the preparatory stage necessary for the contact problem solution has been executed.

Further it is necessary to create final 3D FE mesh of interacting bodies for what generated earlier Plot Only elements are necessary for rotating around vertical axis with the use of the command Mesh Revolve Element when creating the set amount of three-dimensional elements. FE models of considered bodies are created at rotation on 360, but in initial radial section the model is cut. There are node pairs which should be merged by means of the command Tools Check Coincident Nodes. An example of the FE mesh of a test problem is shown on Fig. 3, a at which there are 12 equal sectors on circular coordinate, i.e. the angular width of each sector is equal 30 degrees. This is one more essential difference from the previous planar solution. There is a question of how many sectors are necessary to set for the reception of maximal exact solution of the problem? For the planar test problem such question could not arise. Thus the approach “the more - the better is“ not correct. First, it is still necessary to prove that with the increase in amount of sectors the accuracy of the solution increases, and secondly, the time of the problem solution considerably increases, that essentiality prevents the efficiency of carrying out numerical experiments.

Certainly, for reduction in considered number degrees of freedom the problem it is possible to consider not full bodies, as in Fig. 3, a, but a separate sector, formed by the borders of two radial sections. But here, unfortunately, it is necessary to confront with the program restriction. In the nodes belonging to the planes of sections the boundary conditions should be set, which should limit displacement in normal direction of the section plane. It is obvious, that it can be carried out in the local coordinates connected with section plane. In program MSC.MARC such opportunity exists. However, the used version of the program does not allow to set boundary conditions for contact nodes in local coordinates and to set them in global coordinates it is possible only for the angle $\varsigma = 90$. In this case the planes of sections coincide with main coordinate planes XOY and ZOY. An example of FE meshes generation at such approach is shown on Fig. 3, b. It is necessary to note, that there is an opportunity of artificial enough technique application of the rejected part modeling action by means of the additional bodies set, but this technique has many lacks. As the result for dense meshes in the near contact areas and small sectors on angular width another technique of FE mesh creation was used.

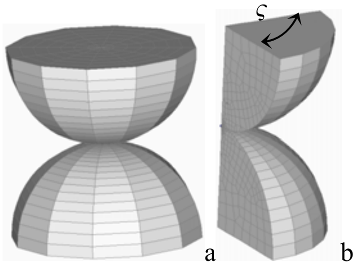

Fig. 3 Examples of spatial FE meshes used for the solution of the test problem

Such technique of the problem solution is rather labour-consuming as instead of setting of two boundary conditions for the top and bottom side that has been described in the beginning of the article, it is necessary to set five boundary conditions (Fig. 4). Thus, for the nodes laying in plane XOY the boundary condition $w_i = 0$, for nodes in plane YOZ - $u_i = 0$, and for nodes on axis OY - both specified conditions should be set. The comparison of solutions for the FE meshes with identical FE digitization in section and identical angular width of separate sectors was made. It is proved, that without dependence on the problem solution technique whether there will be its full FE mesh (Fig. 3, a) or the cut out quarter (ς=90) - (Fig. 3, b) the solution will not change. On Fig. 5 examples of such solutions, the distributions of normal contact stress σ_{yy} are shown.

As we see from the resulted distributions, the con

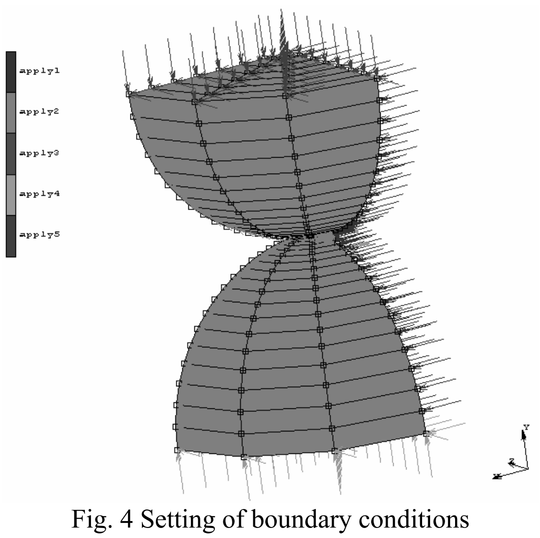

Fig. 4 Setting of boundary conditions

tact zone has the expected form - a correct polygon. It is obvious, that at increase in amount of sectors this form will come nearer to circular form. The normal contact stress as follows from Hertz – Beliaev theory is distributed in conformity with the following dependence

$$\sigma_{yy}^{H}\left(x,y\right)=\begin{cases}\sigma_{max}^{H}\sqrt{1-\left(\dfrac{x}{a}\right)^{2}-\left(\dfrac{y}{b}\right)^{2}}\,; & \left(\dfrac{x}{a}\right)^{2}+\left(\dfrac{y}{b}\right)^{2}\leq 1\\ 0\quad ; & \left(\dfrac{x}{a}\right)^{2}+\left(\dfrac{y}{b}\right)^{2}>1\end{cases} \quad (5)$$

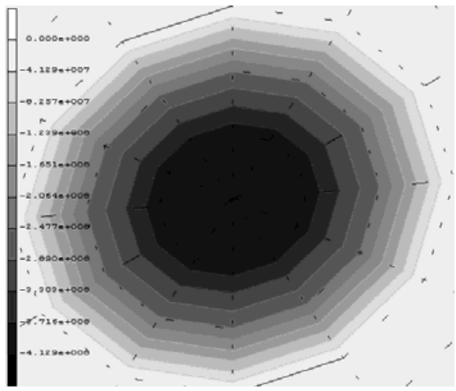

Fig. 5 An example of the distribution of normal contact stress for various FE digitizations

3. Choice of error estimation criterion of the solution

In the Table 1 the results of numerical experiment and Hertz theoretical calculation are compared. We shall note that at this comparison the following approach was used. For set FE digitizations (column 1 - dimension of mesh in near contact zone 1) and set approach Δ a problem has been solved which allowed defining various characteristics including contact nodal forces. It was possible to define unknown in advance total compression force F for the considered bodies by summarizing the specified forces. This procedure is labour-consuming enough, but the program MSC.MARC possesses any additional opportunities which allow the user to improve many algorithms. For example, by means of writing on algorithmic language FORTRAN separate user modules it is possible to set the properties of the finite elements, which have been not stipulated by the basic program. Module UPSTNO.F allows the user at the work with the postprocessor to output additional vectors of the nodal characteristics.

That routine operation of summation of nodal forces was carried out by a computer, module UPSTNO.F has been modified. Thus, the auxiliary file to which at the vector of additional information creation addressed MARC each time and in which the sums of projections of nodal forces collected was used. After the calculation of FEM problem it was enough to read the value of force F in the specified auxiliary file. It is obvious that projections of total force to two other axes are equal to zero, but modified module UPSTNO.F allows to define them which will be used further for the solution of contact problems of interaction between a wheel and a rail. The certain values of the force F are resulted in column 5.

If to consider as the most exact solution which will be looked at below, $F = 2080$ N, the solution received for FE mesh with digitizations in zone 1 of 7x7 elements

Table 1
Comparison of solutions for coordinated FE meshes
(a spatial test problem)

1	2	3	4	5	6	7
1x1	60	1	1	1599	18.781	44.309
	30		1	1615	29.178	50.867
	15		1	1620	31.466	52.341
2x2	60	2	7	2713	11.825	33.544
	30		13	2231	17.269	38.772
	15		8(25)	692(2768)	25.334	43.25
3x3	60	3	13	2064	9.273	13.582
	30		25	2037	3.435	13.499
	15		15(49)	541(2163)	8.185	18.305
4x4	60	3	13	2029	9.802	14.538
	30		25	2065	3.93	8.149
	15		15(49)	519(2074)	8.986	10.832
5x5	60	4	19	2203	7.881	13.53
	30		13(37)	545(2187)	4.773	9.527
	15		22(73)	538(2150)	7.077	11.503
6x6	60	5	25	1991	14.314	26.327
	30		15(49)	527(2098)	1.085	8.948
	15		29(97)	510(2041)	3.124	9.355
7x7	60	5	25	2040	14.233	17.711
	30		15(49)	520(2080)	0.029	4.665
	15		29(97)	523(2091)	3.089	4.957

Notes to Table 1:
Column 1 - dimension of the mesh in zone 1;
Column 2 - angular width of sector in degrees;
Column 3 - number of nodes on radius being contact after loading;
Column 4 - total number of nodes in the contact zone; if the additional number in brackets is resulted. It means that the problem for FE digitizations according to Fig. 3, b was solved, and the number specified in brackets shows how many nodes would be in contact zone if full FE digitization according to Fig. 3, a was considered;
Column 5 - calculated using FEM total compressing force F , N, in case of presence of additional number in the brackets see the previous note;
Column 6 – the error of calculations, % with the formula (6);
Column 7 – the error of calculations, % with the formula (7).

and angular width of 30 degrees sector it appears that the size of defined force F in the big degree depends on FE digitizations. In calculations it changed from 1599 N up to 2768 N, i.e. the error of force definition reached 33 %. And it occurred for coordinated FE mesh that once again proves the necessity of careful choice of FE digitization. It is obvious also, that with the increase in FE meshes density in the near contact areas, the error of force F definition decreased.

Depending of the found force F magnitude with the help of formula (3) the maximal Hertz stress is defined. Specified stress can be compared with the maximal contact stress σ_{yy}^{max} received with the help of FEM for what the following formula is used

$$\delta = \frac{\left|\sigma_{max}^{H} - \sigma_{yy}^{max}\right|}{\sigma_{max}^{H}} \times 100\% \qquad (6)$$

The solution error calculated in the given way is resulted in column 6. Unfortunately, the conclusions made earlier concerning the given method of errors calculation in the solution remains fair and in a spatial case. If to compare calculations for the same FE digitizations in radial section (in zone 1 mesh consists of 7x7 elements), for different angular width of sector 30 or 60 we receive an error 0.029 or 14.233, i.e. the accuracy of calculations differs in 490 times! It is quite obvious that it is not so. Calculations, certainly, differ. It can be judged, for example, on force F . But if to compare the nodal forces difference makes all about 2%. We come to the necessity of using more adequate method of the solution error estimation.

It is obvious, that it is possible to offer many variants of the solution error definition, but it is represented expedient to use other formula

$$\delta = \left| \frac{\int_0^{2a} \left[\sigma_{yy}^{H}(x,0) - \sigma_{yy}(x) \right] dx}{\int_0^{a} \sigma_{yy}^{H}(x,0) dx} \right| \times 100\% \qquad (7)$$

where Hertz contact stresses $\sigma_{yy}^{H}(x,0)$ are calculated with the use of formula (5) and function $\sigma_{yy}(x)$ turns out by means of linear interpolation calculated with contact stresses in the nodes, for which after deformation $x \geq 0,\ y = 0,\ z = 0$. For the calculation of an error under the formula (7) the package MathCAD was used.

The errors of numerical calculation certainly in a greater degree allow to estimate the efficiency of the FE digitization. The analysis of errors for various FE meshes has shown, that there is a general tendency of the decrease of calculation error at the increase in the density of coordinated meshes. But here it is necessary to choose optimum density since with its growth also the time of calculations essentially grows. On the other hand, the increase in amount of sectors in a circular direction is not effective. Moreover, at small angular width of the sector the error starts to grow. The calculations were carried not only for the angles specified in Table 1, but also for smaller ones. The optimum there was an angular width of the sector equal to 30 degrees. This width has been chosen for the influence analysis of non-coordination of FE meshes. Then is it possible to explain this fact? It was already marked earlier that the elongation of finite elements influences negatively on the accuracy of numerical solution. In program MSC.MARC there is an opportunity of checking the elongation of finitel elements. Parameter Aspect Ratio (AR) which is calculated for planar elements as the ratio of the perimeter of an element and its area, and for spatial as the ratio of the area of an element and its volume is supervised. The received ratios are normalized so that for ideal elements on elongation (equipotential triangles, squares, ideal tetrahedrons or cubes) AR=1. Obviously, the more the form of an element differs from ideal, the greater value has AR for the considered element. For linear elements the value AR>3 is considered inadmissible, i.e. carried to the big error of the solution. For parabolic elements such border above AR> 10. If now to consider three solutions (tab. 1) at digitization of the near contact zones 4x4 for various angular width of sectors (15, 30 and 60 degree) the clear advantage on accuracy for calculation with angular width of the sector 30 is available. If to check now how many elements are excessively extended (AR>3) it appears, that for a corner 60 such elements are 348, for a corner 30 are 24, for a corner 15 are also 24. But from the view that for only cut out quarter of a FE mesh the full grid of such elements would be 96 the latter case is considered. Thus, it becomes obvious why FE meshes for angular width of the sector equal to 30 degrees possess advantage of the solution accuracy. The analysis for others FE digitizations has confirmed this conclusion.

In connection with that the influence of mesh non-coordination has been considered in details in work [3], for the spatial test the problem of the influence of non-coordination of nodes has been considered only in a radial direction. It is obvious that the consideration of non-coordination in circular direction or generally is not a serious problem.

The results of such analysis are presented in Table 2 where the data resulted in each column corresponds to the previous Table 1. As base numerical experiment calculation for mesh 7x7 a corner of 30 degrees (a penultimate line of Table 2) was taken. For the presentation these data are repeated in the first line of Table 2. The problem with angular width of the sector equal to 30 degrees and the FE mesh of the bottom body which has remained the same (7x7) is considered alternatively. The mesh is taken from the top body even more dense - 8x8. The calculated compressing force F is practically identical to the considered solution. But the here distribution of contact stress differs essentially. If to consider the base solution more exactly the stress acting on the bottom body (the second line of the Table 2) appear underestimated and the stress acting on the top body (last line) appears overestimated. Accordingly, the error of the solution for non-coordinated mesh on radius increases from 1.5 (for the bottom body) up to 2 times (for the top body).

Table 2

Comparison of solution for coordinated and non-coordinated FE meshes (the spatial test problem)

<table>
<tr><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th><th>6</th><th>7</th></tr>
<tr><td>7x7</td><td rowspan="3">30</td><td>5</td><td>17(49)</td><td>520(2080)</td><td>0.029</td><td>4.665</td></tr>
<tr><td>bottom 7x7</td><td>5</td><td>17(49)</td><td rowspan="2">520(2078)</td><td>4.795</td><td>7.125</td></tr>
<tr><td>top 8x8</td><td>6</td><td>21(61)</td><td>12.744</td><td>9.573</td></tr>
</table>

4. Conclusions

Thus, as the result of carried research it is possible to approve that the developed technique of the solution of contact problems allowing to solve them with high enough accuracy. The recommendation resulted above can be used at the analysis of contact interaction between a wheel and a rail.

References

1. **Sładkowski A., Kuminek T.** Методика моделирования контактного взаимодействия в паре колесо – рельс // XIV научна конференция с международно участие «Транспорт 2004».-София: Висше транспортно училище «Тодор Каблешков», 2004, с.237-240.
2. **Sładkowski, A., Sitarz, M.** Analysis of wheel-rail interaction using FE software.-Wear, 2005, No258, c.1217-1223.
3. **Sładkowski, A., Kuminek, T.** Influence of the FE discretization on accuracy of calculation of contact stress in a system wheel – rail.-Proc. 3rd Sc. Conf. of Jan Perner Transport Faculty “New Trends in Transport and Communications”.-Pardubice: University of Pardubice, 2003, p.13-18.
4. **Пономарев С.Д., Бидерман В.Л., Лихачев К.К. и др.** Расчеты на прочность в машиностроении. -Москва: Машгиз, 1958, т.2.-974с.

A. Sładkowski

ERDVINIŲ KONTAKTINIŲ UŽDAVINIŲ SPRENDIMO BEM TIKSLUMO ANALIZĖ

R e z i u m ė

Darbe išnagrinėta tamprių pussferių kontaktinė sąveika, išreiškiama Herco ir Beliajevo sprendiniu. Rastas šio uždavinio sprendimas BEM, nustatytos sprendimo paklaidos esant skirtingiems BE tinkleliams. Sukurta pusiau automatinio BE tinklelių generavimo metodika, leidžianti sumažinti uždavinio sprendimo paklaidą.

A. Sładkowski

ACCURACY ANALYSIS OF THE SOLUTION OF SPATIAL CONTACT PROBLEM BY MEANS OF THE FEM

S u m m a r y

In the work was a solved contact interaction problem of elastic hemispheres which has the form of Hertz – Beliaev solution. The solution of the same problem by means of FEM is simultaneously examined. Errors of the solution for various FE meshes are determined. Techniques of semi automatic generation of the FE meshes for reducing an error of the problem □educetion are developed.

А. Сладковский

АНАЛИЗ ТОЧНОСТИ РЕШЕНИЯ ПРОСТРАНСТВЕННЫХ КОНТАКТНЫХ ЗАДАЧ ПРИ ПОМОЩИ МКЭ

Р е з ю м е

В работе рассмотрено контактное взаимодействие упругих полусфер, для которого существует решение Герца – Беляева. Одновременно найдено решение указанной задачи при помощи МКЭ. Определены погрешности решения для различных КЭ сеток. Разработана методика полуавтоматической генерации КЭ сеток, которая позволяет уменьшить погрешность решения задач.

Received December 22, 2004