

COMPARISON OF SOME CONTACT ELEMENTS AND ITERATIVE ALGORITHMS FOR THE SOLUTION OF FRICTIONLESS CONTACT PROBLEMS

JÁNOS ÉGERT

Department of Mechanics and Machine Structures, Széchenyi István College Győr,
5 Kovács Pál Str., 9021 GYŐR, Hungary
egert@rs1.szif.hu

[Received: August 15, 1999]

Abstract. The paper presents three different iterative algorithms for the numerical solution of contact problems. Each algorithm is based on the use of the finite element method. It is assumed that the contact conditions are fulfilled by means of special contact elements. Numerical results for the various contact elements and solution algorithms are then compared by solving 2D elasticity problems.

Keywords: Iterative algorithms, frictionless contact problems, finite element method

1. Introduction

A machine structure consists of several parts, which are connected with each other by surface contact. It is very important to determine the displacements, strains and stresses in the contact area, because they can exert a powerful influence on the reliability, failsafety and lifetime of the construction.

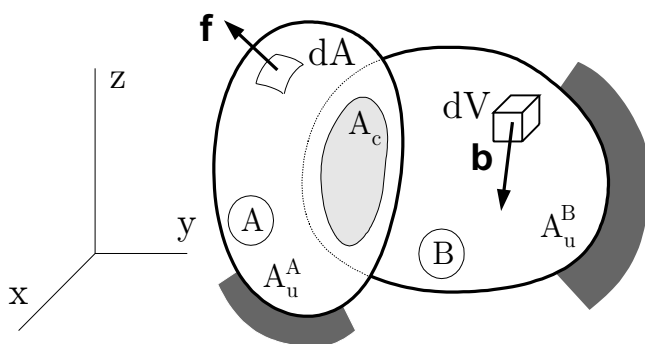


Figure 1.

An elastic contact problem can be formulated as follows. The system under consideration – see Figure 1 for details – consists of two bodies denoted by \mathcal{A} and \mathcal{B} . These bodies are in contact on the surface A_c . The body forces \mathbf{b} and surface tractions \mathbf{f} on $A_t^{\mathcal{A}}$ and $A_t^{\mathcal{B}}$ are given for each body. The kinematic boundary conditions (support conditions) are also known on the corresponding surfaces $A_u^{\mathcal{A}}$ and

$A_u^{\mathcal{B}}$. ($V = V^{\mathcal{A}} \cup V^{\mathcal{B}}$; $V^{\mathcal{A}} \cap V^{\mathcal{B}} = 0$; $A^{\mathcal{A}} = A_c^{\mathcal{A}} \cup A_u^{\mathcal{A}} \cup A_t^{\mathcal{A}}$; $A^{\mathcal{B}} = A_c^{\mathcal{B}} \cup A_u^{\mathcal{B}} \cup A_t^{\mathcal{B}}$; $A_t = A_t^{\mathcal{A}} \cup A_t^{\mathcal{B}}$). Though it is assumed that the equations of linear elasticity are valid for each body (the material is homogenous and isotropic) due to the surface contact the relation between loading and displacements is a non-linear one. Consequently the problem is also non-linear.

For non-linear problems the incremental version of the principle of virtual work is the basis of solution. For the structure made up of the two bodies \mathcal{A} and \mathcal{B} , the principle is of the form

$$\int_{(V)} \Delta \mathbf{T} \cdot \cdot \delta(\Delta \mathbf{E}) dV = \int_{(V)} \Delta \mathbf{b} \cdot \delta(\Delta \mathbf{u}) dV + \int_{(A_t)} \Delta \mathbf{f} \cdot \delta(\Delta \mathbf{u}) dV . \quad (1.1a)$$

In addition it holds for body \mathcal{A} (and \mathcal{B}) that

$$\begin{aligned} \int_{(V^{\mathcal{A}})} \Delta \mathbf{T} \cdot \cdot \delta(\Delta \mathbf{E}) dV + \int_{(A_c)} \Delta \mathbf{p} \cdot \delta(\Delta \mathbf{u}) dV = \\ \int_{(V^{\mathcal{A}})} \Delta \mathbf{b} \cdot \delta(\Delta \mathbf{u}) dV + \int_{(A_t^{\mathcal{A}})} \Delta \mathbf{f} \cdot \delta(\Delta \mathbf{u}) dA \end{aligned} \quad (1.1b)$$

where \mathbf{T} is the stress tensor, \mathbf{E} is the strain tensor, \mathbf{p} is the contact pressure, \mathbf{u} is the displacement field, the double scalar product is denoted by $\cdot \cdot$. Greek δ stands for the variation of a quantity while Δ identifies the increment of a quantity.

There are two typical groups of solution methods used to solve elastic contact problems of the above type. Methods in the first group reduce the contact problem to a mathematical programming problem [3] [4]. Procedures in the second group are iterative methods based on the use of special contact elements [5] - [9]. This paper is going to apply and compare three different iterative methods and three types of contact elements for solving frictionless elastic contact problems. The main goal of the author was to implement two known iterative methods [5] - [9] and three types of contact elements into a FEM code and to compare those with a new iterative solution algorithm.

With the iterative solutions the contact conditions for each increment and iteration step are satisfied using special contact elements. These special contact elements – see Figure 2 – are fictitious elements with proper stiffness or material properties which ensure the fulfillment of contact conditions. Thus they are not real finite elements, but can be considered as an appropriate numerical help for the solution process.

In order to fulfil the contact condition in each load increment, two known iterative solution algorithms were implemented: iteration with varying stiffness, iteration with constant stiffness and a new method was developed: iteration with kinematic load.

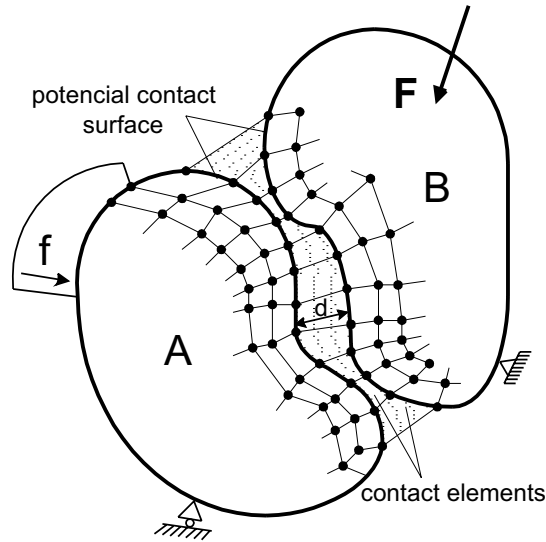


Figure 2.

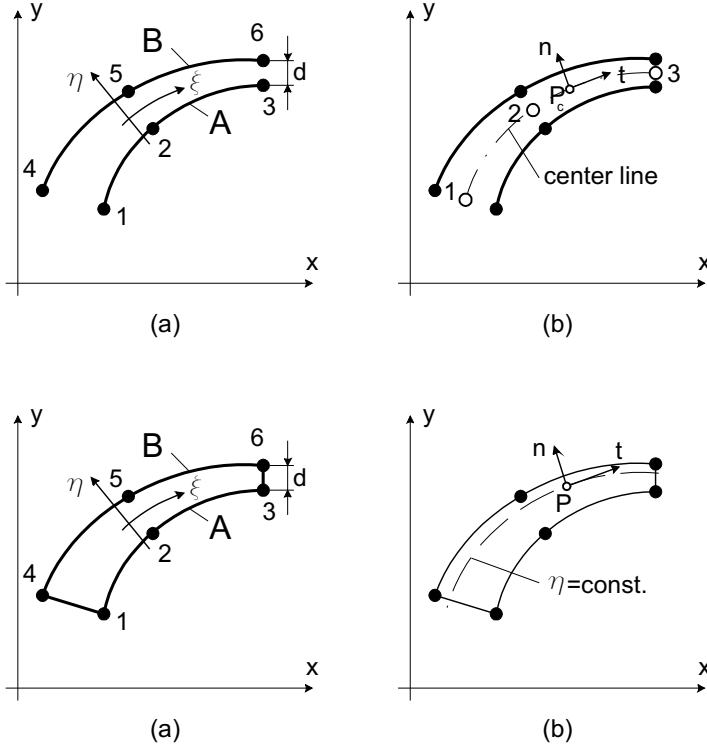
Every contact elements and solution algorithms was installed in the *FEM-3D* finite element program system [13] developed under *Páczelt's* leadership. The effective application of the above iterative algorithms with contact elements assumes that the FEM code makes substructuring possible. During the solution of contact problems it is very useful to select the nodes on the potential contact area (Figure 2) as external nodes while the other nodes can be considered as internal nodes. In this case one should carry out the iteration steps taking only the variables of external nodes into consideration.

2. Features of 2D contact elements

Three types of contact elements, namely line contact elements, layer contact elements and node (or spring) contact elements were applied during the computations. These special elements have already been applied in the literature [5] - [9]. Figures 3-5 illustrate the elements for 2D case.

The line contact elements (Figure 3) consist of two lines which represent real element sides on bodies *A* and *B* and are attached to each other via a distributed spring system. The corresponding spring stiffnesses are denoted by k_t and k_n in the local co-ordinate system t, n (Figure 3/b.).

During the contact iteration the stiffnesses k_t and k_n must be modified until the tangential (frictional) and normal contact conditions are fulfilled. For the characteris



Figures 3 and 4

tic block of stiffness matrix of line contact element we obtain

$$\mathbf{K}_{c_{ij}} = \begin{bmatrix} k_t & 0 \\ 0 & k_n \end{bmatrix}_{ij} \tag{2.2}$$

where the nodes attached to each other by the spring are denoted by i and j .

When using line contact elements the fulfillment of contact conditions is ensured in the tn co-ordinate system along the center line of the contact element.

The layer contact elements (Figure 4) are made of special fictitious material for which the modulus of elasticity E and the shear modulus G do not depend on each other. Fulfillment of the normal contact condition can be reached by modifying E and by giving a proper value to the shear modulus G one can satisfy the frictional contact conditions. The form of the layer contact element stiffness matrix is similar to that expressed in equation (1.1b). The contact conditions are again fulfilled along the $\eta = 0$ co-ordinate line.

The node contact elements (Figure 5) consist of a couple of nodes, which are connected in tangential and normal direction by springs with stiffnesses k_t and k_n .

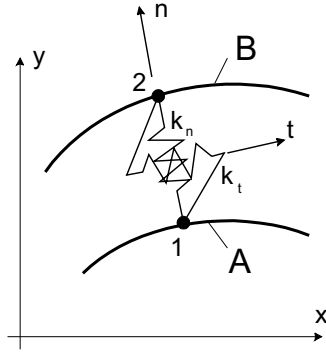


Figure 5.

The contact iteration procedure is the same as for the case of the above two contact elements, consequently the stiffness matrix has the same form with only one characteristic block.

3. Iterative solution algorithms

The basic FEM equation of the investigated system of bodies A and B for the i -th load increment $\Delta \mathbf{f}^i$ may be written as

$$(\mathbf{K}_{AB} + \mathbf{K}_c) \Delta \mathbf{q}^i = \Delta \mathbf{f}^i \quad (3.1)$$

where \mathbf{K}_{AB} is the super-structure stiffness matrix of the bodies A and B and \mathbf{K}_c is the stiffness matrix of the contact elements. If \mathbf{K}_c takes the "proper" value, (i.e., each contact condition is fulfilled), these (the above) equations will allow for the computation of the real displacement increment of the external nodes. It follows that our task is now to determine this "proper" value for \mathbf{K}_c . For this purpose, three iterative solution methods have been developed:

a) *Iteration with varying stiffness* [5] - [8]

In the 1-st step we solve the linear algebraic equation system (3.1) with an initial value \mathbf{K}_{c_0} :

$$\Delta \mathbf{q}_0^i = (\mathbf{K}_{AB} + \mathbf{K}_{c_0})^{-1} \Delta \mathbf{f}^i. \quad (3.2)$$

Then the fulfillment of contact conditions is checked for the solution $\Delta \mathbf{q}_0^i$ and in case they are not fulfilled, the stiffness matrix of contact elements must be modified by \mathbf{K}_{c_1} - as regards its choice see [12] - :

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_0} + \mathbf{K}_{c_1}) \Delta \mathbf{q}_1^i = \Delta \mathbf{f}^i \quad (3.3)$$

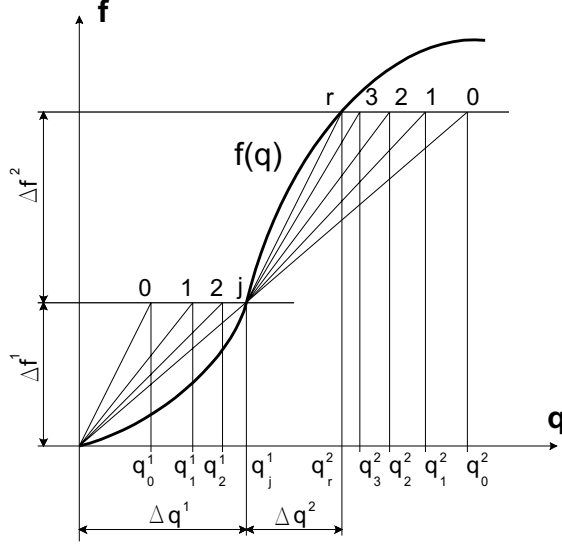


Figure 6.

After solving equation system (3.3) we obtain a new approach for the nodal displacement increment:

$$\Delta \mathbf{q}_1^i = (\mathbf{K}_{AB} + \mathbf{K}_{c_o} + \mathbf{K}_{c_1})^{-1} \Delta \mathbf{f}^i . \quad (3.4)$$

In the $(j+1)$ -th iteration step the displacement increment can be computed as follows:

$$\Delta \mathbf{q}_j^i = (\mathbf{K}_{AB} + \mathbf{K}_{c_o} + \mathbf{K}_{c_1} + \dots + \mathbf{K}_{c_j})^{-1} \Delta \mathbf{f}^i . \quad (3.5)$$

From this equation it is obvious that the iteration algorithm with varying stiffness needs a new matrix inversion procedure in each iteration step since the stiffness of the contact elements is changed step by step. The qualitative character of the above procedure is shown in Figure 6.

b) Iteration with constant stiffness [9]

The 1-st step is the same as for the case of the iteration with varying stiffness – see equation (3.2). Rearranging equation (3.3) we obtain a new equation with an unchanged left side system matrix (stiffness matrix):

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_o}) \Delta \mathbf{q}_1^i = \Delta \mathbf{f}^i - \mathbf{K}_{c_1} \Delta \mathbf{q}_0^i . \quad (3.6)$$

Introducing a new notation for the right side of the above linear equation system

$$\Delta \mathbf{f}_1^i = \Delta \mathbf{f}^i - \mathbf{K}_{c_1} \Delta \mathbf{q}_0^i , \quad (3.7)$$

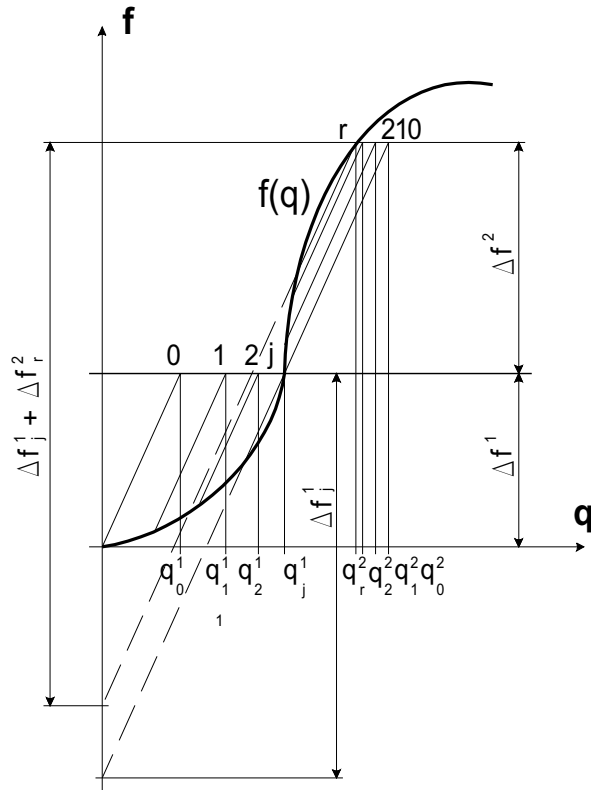


Figure 7.

a recursive formula with constant stiffness matrix and varying load increment is obtained:

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_o}) \Delta \mathbf{q}_1^i = \Delta \mathbf{f}_1^i \quad (3.8)$$

For the $(j + 1)$ -th iteration step the displacement increment can be computed from the following equation:

$$\Delta \mathbf{q}_j^i = (\mathbf{K}_{AB} + \mathbf{K}_{c_o})^{-1} \Delta \mathbf{f}_j^i \quad (3.9)$$

where the current value of load increment is

$$\Delta \mathbf{f}_j^i = \Delta \mathbf{f}^i - \mathbf{K}_{c_1} \Delta \mathbf{q}_0^i - \dots - \mathbf{K}_{c_j} \Delta \mathbf{q}_{(j+1)}^i \quad (3.10)$$

It is seen from equation (3.10) that the iteration procedure with constant stiffness requires only one matrix inversion for each load increment and that, step by step, only the right side is modified. In this manner, it is possible to save a great deal of computing operations and time. Figure 7. represents the qualitative character of the above algorithm.

c) Iteration with kinematic load

This algorithm is based on *Páczelt's* idea [14]. Using this idea an appropriate kinematic loading is applied in the 1-st iteration step:

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_o}) \Delta \mathbf{q}_0^i = \Delta \mathbf{f}_0^i + \mathbf{f}_{g_o} \quad (3.11)$$

The kinematic loading can be computed as

$$\mathbf{f}_{g_o} = \mathbf{K}_{c_o} \mathbf{g}_0 \quad (3.12)$$

where \mathbf{g}_0 is the matrix of the initial gaps of normal direction on the potential contact area. This kinematic loading is set to a value which makes the initial normal gaps disappear.

Solving the algebraic equation (3.11) one can check the fulfillment of the contact condition and after performing the necessary modifications in the contact stiffness matrix a new equation system is obtained:

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_o} + \mathbf{K}_{c_1}) \Delta \mathbf{q}_1^i = \Delta \mathbf{f}^i + (\mathbf{K}_{c_o} + \mathbf{K}_{c_1}) \mathbf{g}_0 \quad (3.13)$$

In the $(j + 1)$ -th iteration step the i -th displacement increment is computed from the following equation system:

$$(\mathbf{K}_{AB} + \mathbf{K}_{c_o} + \dots + \mathbf{K}_{c_j}) \Delta \mathbf{q}_j^i = \Delta \mathbf{f}^i + (\mathbf{K}_{c_o} + \dots + \mathbf{K}_{c_j}) \mathbf{g}_0 \quad (3.14)$$

It is seen from equation (3.14) that both the stiffness matrix and the load vector are modified step by step in this iteration procedure.

4. Numerical examples

The comparison of efficiency and capability of the above algorithms and contact elements are carried out through the use of two numerical tests.

a) Elastic cylinder - rigid plane

In this example we have solved the contact problem of an infinite elastic cylinder and a rigid plane pressed to each other by a uniform load \mathbf{q} (Figure 8). Thus the problem is a plane strain one.

H. Hertz [15] found an exact solution (closed form solution) for this frictionless contact problem, which allows us to compare our results with the exact solution. Taking advantage of the symmetrical properties of the problem, as a finite element model we can regard half a cylinder loaded by half of the given loading. The top part of the cylinder does not influence the behavior of the contact area, therefore we are allowed to neglect the top quarter of our half cylinder. Figure 9 shows the FEM model of the investigated problem.

In Figure 10 we have compared Hertz's exact solution with various approximate solutions – see Figure 10 for the details concerning the methods of the approximate so-

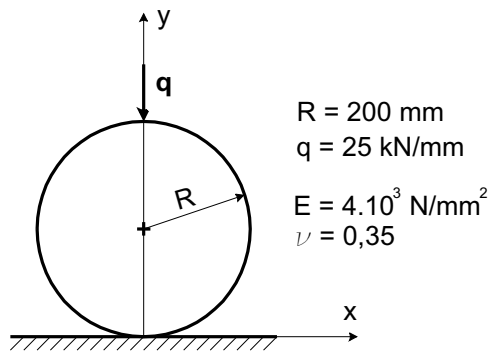


Figure 8.

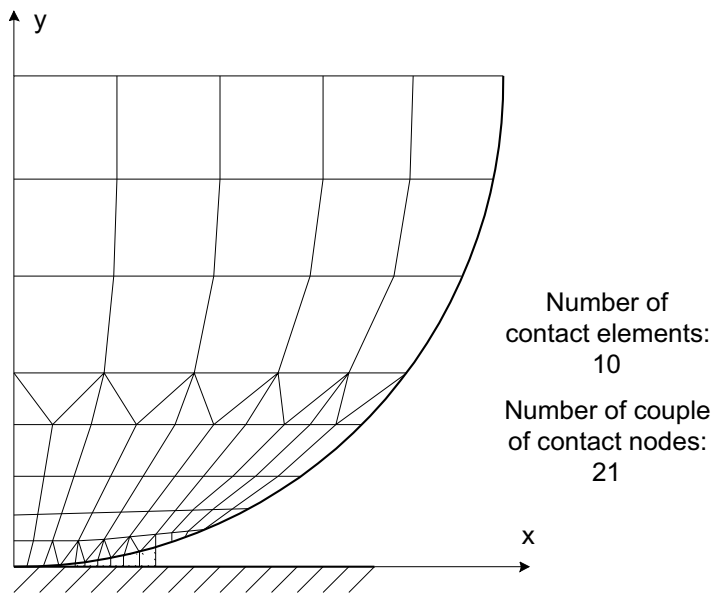


Figure 9.

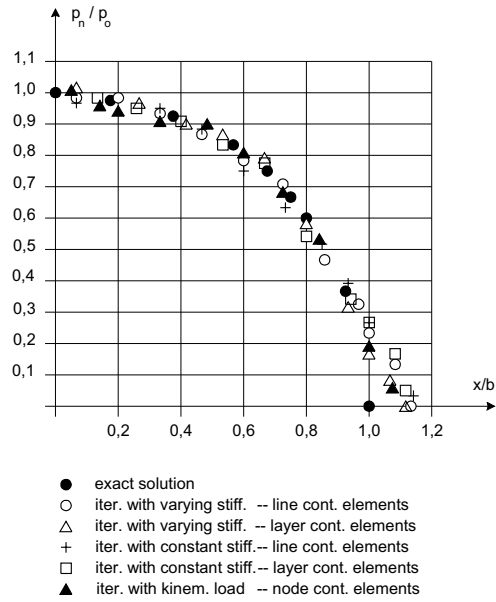


Figure 10.

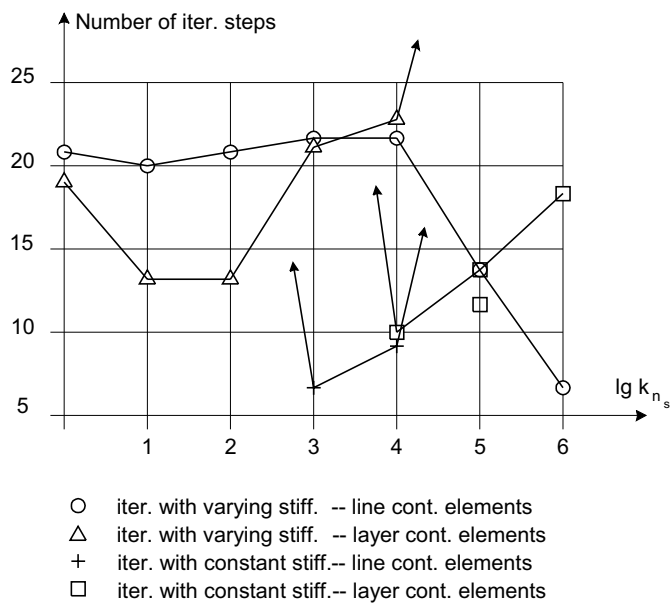


Figure 11.

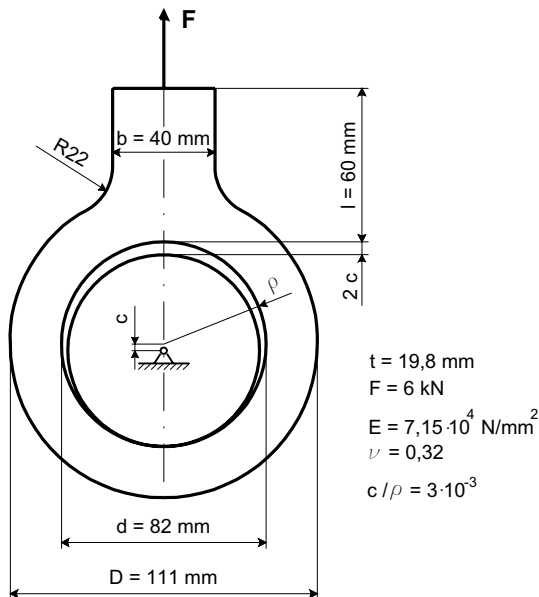


Figure 12.

lutions. It can be seen that we have obtained a good agreement for every combination of the applied contact elements and iterative algorithms.

At the same time, the number of iteration steps required to reach convergence has differed to a great extent, and it has strongly depended on the initial value of the normal stiffnesses of contact elements (Figure 11). In some cases there was no convergence found at all. From Figure 11 it is seen that the iteration with varying stiffness is much more stable than the iteration with constant stiffness. For the iteration with varying stiffness and using line contact elements, convergence was found in every case. The best results are obtained using the iteration with kinematic load. They are not seen in Figure 11, because with this iterative algorithm the convergence was reached in two steps for any initial normal stiffness.

b) Pin - connecting rod problem

In this problem, (Figure 12) a connecting rod acts on a supported pin with a radial eccentricity c . Both the pin and the rod (housing) are assumed to have the same thickness t . Thus it is, from a mechanical point of view, a plane stress problem.

The load is a concentrated force \mathbf{F} exerted on the rod. Figure 13. shows the finite element model which takes advantage of the symmetry of both parts of the joint.

In Figures 14 and 15, solutions obtained using the presented solution methods are compared with each other and with Gaertner's FEM and experimental results [16]. Both figures show a reasonable agreement in computation accuracy of the different solutions.

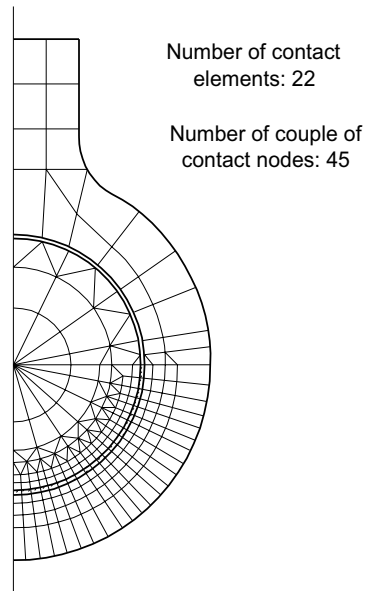


Figure 13.

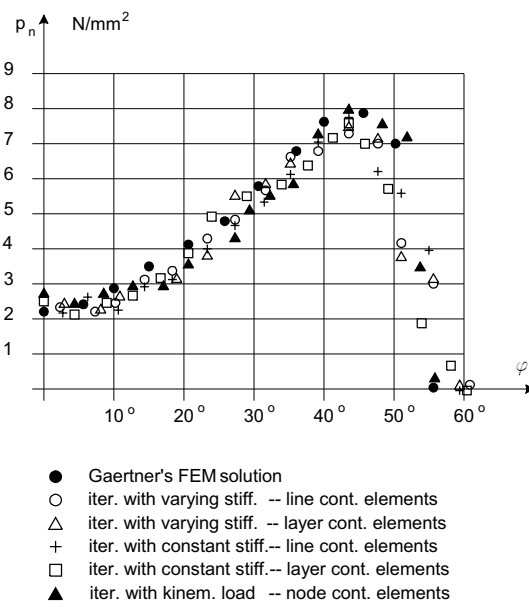


Figure 14.

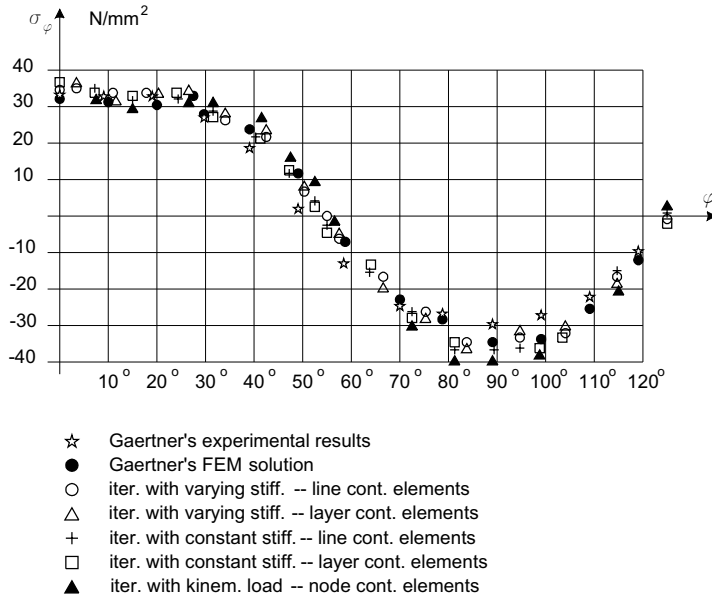


Figure 15.

For the latter problem the number of iteration steps required to reach convergence depends on the initial value of the normal stiffnesses of the contact elements. The applied algorithms and contact elements have shown the same convergence properties as for the first sample problem. When solving this problem we obtained the least iteration steps using an iteration algorithm with kinematic load. This iterative algorithm was convergent in two steps independently of the initial value of normal stiffnesses.

5. Concluding remarks

All of the applied iterative algorithms and contact elements have given a good approximation for the investigated sample problems. Within the prescribed accuracy, there is no remarkable difference between the implemented iterative methods or between the applied contact elements.

But there is a great difference in the number of iteration steps required to reach convergence. As regards the methods taken from the literature the number of iteration steps strongly depends on the initial value of the normal stiffnesses of the contact elements. In some cases there was no convergence found at all. The iteration procedure with varying stiffnesses has proved to be convergent in all cases provided that line contact elements have been used.

The new algorithm suggested in this paper, i.e., the iteration with kinematic load using node contact elements has also proved to be convergent in every case. In

addition it always works independently of the initial value of the normal stiffness. It is a further advantage that the number of iteration steps to be carried out and the computational time are much less than those for the other solution algorithms.

REFERENCES

1. BATHE, K. J.: *Finite Element Procedures*, Upper Saddle River, New Jersey, 1996.
2. KLEIBER, M.: *Handbook of Computational Solid Mechanics - Survey and Comparison of Contemporary Methods*, Springer-Verlag, Berlin Heidelberg New York, 1998.
3. PÁCZELT, I.: *Some remarks to the solution of quadratic programmings problems*, Publications Techn. Univ. Heavy Ind. Miskolc, Ser. D. Natural Sciences, **33**, (1979), 137-156.
4. FISCHER, U., HOYER, U. and MELOSH, R.-J.: *Lösung diskretisierter Kontaktprobleme unter Verwendung der linearen Optimierung*, Technische Mechanik, **8**, (1987), 52-56.
5. KATONA, M. G.: *A simple contact-friction interface element with applications to buried culverts*, Int. Journ. Num. Meth. Geomech., **7**, (1983), 371-384.
6. DESAI, C. S., ZAMAN, M. M., LIGHTNER, J. G. and SIRIWARDANE, H. J.: *Thin-layer element for interfaces and joints*, Int. Journ. Num. Anal. Meth. Geomech., **8**, (1984), 19-43.
7. KEUSER, M.: *Verbundelemente für nichtlineare Finite-Element-Berechnungen vom Stahlbaukonstruktionen*, VDI Fortschrittberichte, Reihe 4., Bauingenieurwesen Nr. 71. VDI Verlag Düsseldorf, 1985.
8. STADTER, J. T. and WEISS, R. O.: *Analysis of contact through finite element gaps*, Comp. & Struct., **10**, (1979), 867-873.
9. GNUTSHI, J. B.: *Solution of contact problems of the theory of elasticity and plasticity*, Problemi prochnosti, **12**, (1982), 99-104.
10. ÉGERT, J. and ALTENBACH, J.: *Inkrementelle Iterationsmethoden zur Lösung elastischer Kontaktaufgaben mit Reibung mittels spezieller isoparametrischer 2D- und 3D Kontaktelemente*, Technische Mechanik, **10**, (1989), 120-137.
11. ALTENBACH, J., ÉGERT, J. and PÁCZELT, I.: *Lösung ebener und rotationssymmetrischer elastischer Kontaktaufgaben mit Hilfe spezieller finiter Kontaktelemente*, Z. angew. Math. Mech. **70**, (1990), T671-T674.
12. ALTENBACH, J., ÉGERT, J. and PÁCZELT, I.: *Vergleich unterschiedlicher Iterationsmethoden zur Lösung elastischer Kontaktaufgaben mit Reibung*, Z. angew. Math. Mech., **72**, (1992), T137-141.
13. ÉGERT, J., HERPAI, B., NÁNDORI, F., PÁCZELT, I., SÁRKÖZI, L. and SZABÓ, T.: *Characteristics of a substructural finite element programming system for IBM AT compatible PC*, Proc. of Int. Conference on Eng. Design (ICED '88), Budapest, 23-25 August 1988, Series WDK 16, Vol. 3.
14. PÁCZELT I.: *Oral communication*, 1990.
15. PONOMAREW, S. D., BIDERMANN, W. L., LIHAREW, K. K., MAKUSHIN, W. M., MALININ, H. H. and FEODOSEW, W. I.: *Rascheti na prochnost w mashinostroenii*, Tom 3., Gl. 6., Mashgiz, Moskva, 1958. (in Russian)
16. GAERTNER, R.: *Investigation of plane elastic contact allowing for friction*, Comp. & Struct., **7**, (1977), 59-63.