Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 3, 2009, vol. LV
article No. 1734

Jiří PODEŠVA*

THE CONTACTS ON THE SUPERELEMENTS

KONTAKTY NA SUPERPRVCÍCH

Abstract

The modeling of the roller bearing brings two important problems. The contact problem means that two surfaces touch one another. The contact transmits the pressure force but not tensile force. It is not known neither the contact area nor the distribution of the contact pressure. The second problem is the largeness of the structure, the number of degrees of freedom (DOF). Nowadays the hardware allows solving the large systems of equations. Nevertheless if the number of DOF goes over 105 or 106 and the algorithm necessitates the iterative approach this could be the problem. The effective way of solution can be domain decomposition using so called super-elements. The paper brings the description of modeling with super-elements and solving the contact problem

Abstrakt

Modelování valivého ložiska přináší dva významné problémy. U kontaktního problému předpokládáme, že dva povrchy se budou dotýkat. Dotyk přenáší tlakovou sílu, ne však tahovou. Není dopředu jasné ani jak velká bude kontaktní plocha, ani jaké bude rozložení kontaktního tlaku. Druhým problémem může být velikost úlohy. Dnešní hardware sice umožňuje rychlé řešení velkých úloh. Přesto při počtu stupňů volnosti v řádu 105 až 106 a při použití algoritmů, vyžadujících opakování výpočtu, může toto představovat problém. Účinnou cestou řešení může být dekompozice modelu na tzv. „super-prvky“. Příspěvek přináší popis modelování s použitím super-prvků a doplnění modelu o kontaktní prvky.

1 INTRODUCTION

The finite element method (FEM) is nowadays widely spread computational method for solving not only several tasks of the mechanics of continuum but also other kinds of physical problems. The application brings the necessity to solve several kinds of accompanying problems.

Nowadays already typical problem of the modeling of the mechanical structures is the contact problem. The two bodies are in contact, one surface simply touches the other. Modeling of such kind of joint brings two problems. The contact is non-linear, transmits the pressure force but not tensile one. If the contact transmits the pressure force not only the distribution of the contact pressure, even the quantity and shape of the contact area are not known. (The Hertz solution is known for a few simple examples, what comes short for solving practical problems.) The commercial program packages for FEM usually offer the tools for solving this. Nevertheless the contact problem is not “closed chapter” and still is a subject of research.

The FEM modeling practice often brings the necessity of solving the extremely large systems of equations. However the quick development of hardware increases the possibilities also another effective ways in the area of mathematics are important. One of these is domain decomposition. The whole area of solution is split into a few smaller sub-areas. The solution then has three phases: the solution of the sub-area matrices, the solution on the sub-area boundaries, the solution in the sub-area interior. For the sub-area the term “super-element” or “macro-element” is usually used.

* doc., Ing., Ph.D., Department of Mechanics, Faculty of Mechanical Engineering, VŠB - Technical University of Ostrava, 17. listopadu 15, Ostrava, Czech Republic, tel. (+420) 59 732 4350, e-mail jiri.podesva@vsb.cz

2 THE CONTACT PROBLEM

The contact problem between two bodies, touching one another, is usually solved by two sets of surface elements, covering the contact surfaces. In the Ansys program they are called “contact elements” and “target elements” (see Fig. 1). One body is covered by contact elements, the other by target elements. During the solution the penetration of the contact surface nodes into the target surface is checked. There are two main methods, the Lagrange method and the penalty method, and the number of auxiliary tools for this.

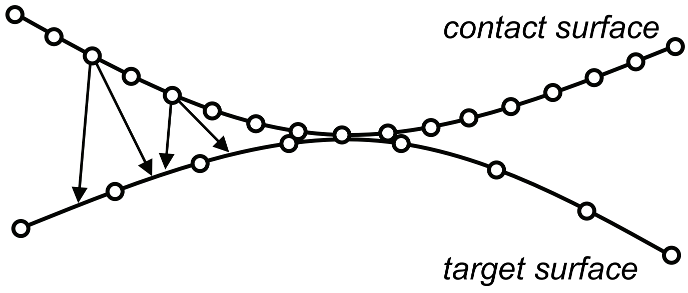

Fig. 1 The contact and the target surface

3 THE DOMAIN DECOMPOSITION

This technique leads to the strong decrease of the number of degrees of freedom (DOF). The methods of reduction can be divided into two groups.

The elimination methods consist in eliminating (neglecting) the large number of DOF. The typical representative is the static condensation method.

The transformation methods consist in defining the totally new set of unknown coordinates (of no physical meaning) using transformation matrix. The typical representative is the modal transformation method.

The domain decomposition method belongs to the first group.

Consider the classic task of the linear static, written in matrix form.

$$\mathbf{K}\cdot\mathbf{q}=\mathbf{f}$$

where $\boldsymbol{K}$ is the stiffness matrix, $\boldsymbol{q}$ is the vector of unknown translations and $\boldsymbol{f}$ is the vector of loading forces. Let us split the original set of DOF $\boldsymbol{q}$ into the sub-set $\boldsymbol{q_m}$ of so called “master” DOF, which will be retained after reduction, and the sub-set $\boldsymbol{q_s}$ of so called “slave” DOF, which will be eliminated. The mathematical record will then be :

$$\begin{bmatrix}\mathbf{K}_{mm} & \mathbf{K}_{ms}\\ \mathbf{K}_{sm} & \mathbf{K}_{ss}\end{bmatrix}\cdot\begin{Bmatrix}\mathbf{q}_m\\ \mathbf{q}_s\end{Bmatrix}=\begin{Bmatrix}\mathbf{f}_m\\ \mathbf{f}_s\end{Bmatrix}$$

or

$$\mathbf{K}_{mm}\cdot\mathbf{q}_m+\mathbf{K}_{ms}\cdot\mathbf{q}_s=\mathbf{f}_m$$
$$\mathbf{K}_{sm}\cdot\mathbf{q}_m+\mathbf{K}_{ss}\cdot\mathbf{q}_s=\mathbf{f}_s$$

If we will derive from the second group of equations:

$$\mathbf{q}_s=\mathbf{K}_{ss}^{-1}\cdot\left(\mathbf{f}_s-\mathbf{K}_{sm}\cdot\mathbf{q}_m\right)$$

or

$$\mathbf{q}_s=\mathbf{K}_{ss}^{-1}\cdot\mathbf{f}_s-\mathbf{K}_{ss}^{-1}\cdot\mathbf{K}_{sm}\cdot\mathbf{q}_m$$

putting into the first group of equations we obtain the system of equations for the master DOF.

$$\left(\mathbf{K}_{mm} - \mathbf{K}_{ms} \cdot \mathbf{K}_{ss}^{-1} \cdot \mathbf{K}_{sm}\right) \cdot \mathbf{q}_m = \mathbf{f}_m - \mathbf{K}_{ms} \cdot \mathbf{K}_{ss}^{-1} \cdot \mathbf{f}_s$$

After substitution:

$$\widetilde{\mathbf{K}} = \mathbf{K}_{mm} - \mathbf{K}_{ms} \cdot \mathbf{K}_{ss}^{-1} \cdot \mathbf{K}_{sm}$$
$$\widetilde{\mathbf{f}} = \mathbf{f}_m - \mathbf{K}_{ms} \cdot \mathbf{K}_{ss}^{-1} \cdot \mathbf{f}_s$$

the equations have the same form as the original equations.

$$\widetilde{\mathbf{K}} \cdot \mathbf{q}_m = \widetilde{\mathbf{f}}$$

Here $\widetilde{\boldsymbol{K}}$ is the reduced stiffness matrix, and $\widetilde{\mathbf{f}}$ is the reduced force vector. The solution can be extended by the reduced mass matrix $\widetilde{\boldsymbol{M}}$ and reduced damping matrix $\widetilde{\boldsymbol{B}}$ into the area of linear dynamics.

To the written above we must add that while the original stiffness matrix $\boldsymbol{K}$ is narrow strip and sparse, the reduced stiffness matrix $\widetilde{\boldsymbol{K}}$ is full. That is why the set of master DOF must be as small as possible.

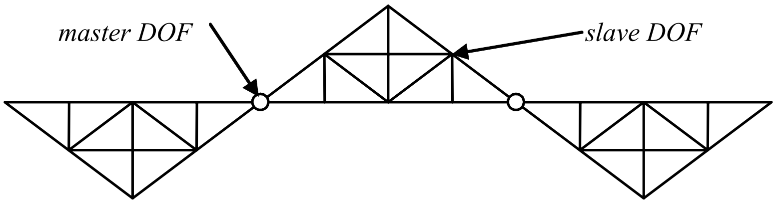

Fig. 2 The main structure divided into three sub-structures

Consider the mechanical structure (see Fig. 2), whose topology offers the natural dividing into a few sub-structures, joined together in boundaries of the very small number of DOF. These will be retained as master DOF, while in the interior of sub-structures are hidden slave DOF.

The reduced stiffness matrix $\widetilde{\boldsymbol{K}}$ of such structure represents the stiffness matrix of the structure in which the single sub-structure seems to be the single finite element. However because in real they are rather large-scale systems they are called “super-elements”.

That is evident from above mentioned that these sub-areas (super-elements) must be internally linear. On the other hand if used to build the larger model, such a model can contain also elements of other types and also non-linearities.

4 THE BEARING MODEL

The subject of modeling is the two row roller bearing of the special design (see Fig 3). The objective is to determine the contact pressure on the rollers. For this reason the standard finite element model was built using the eight-nodes 3D elements (bricks). On the touching surfaces of the inner ring, two outer rings and rollers the contact pair elements were generated. The bearing consists of the inner ring, two outer rings and 70 rollers (in two series, 35 rollers both).

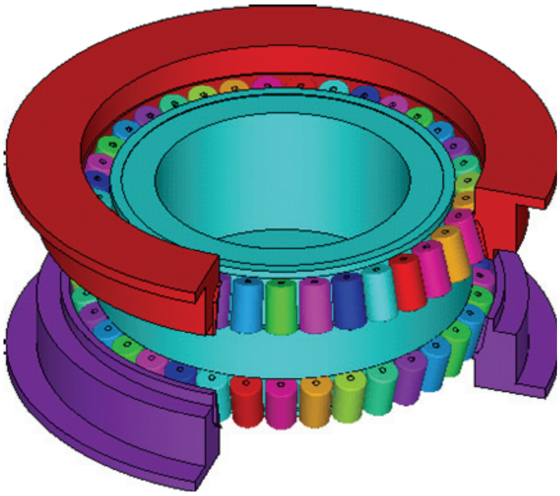

Fig. 3 The bearing

The note: Surely all rings are full (360°). On the figure the outer rings are segmented to see into the bearing.

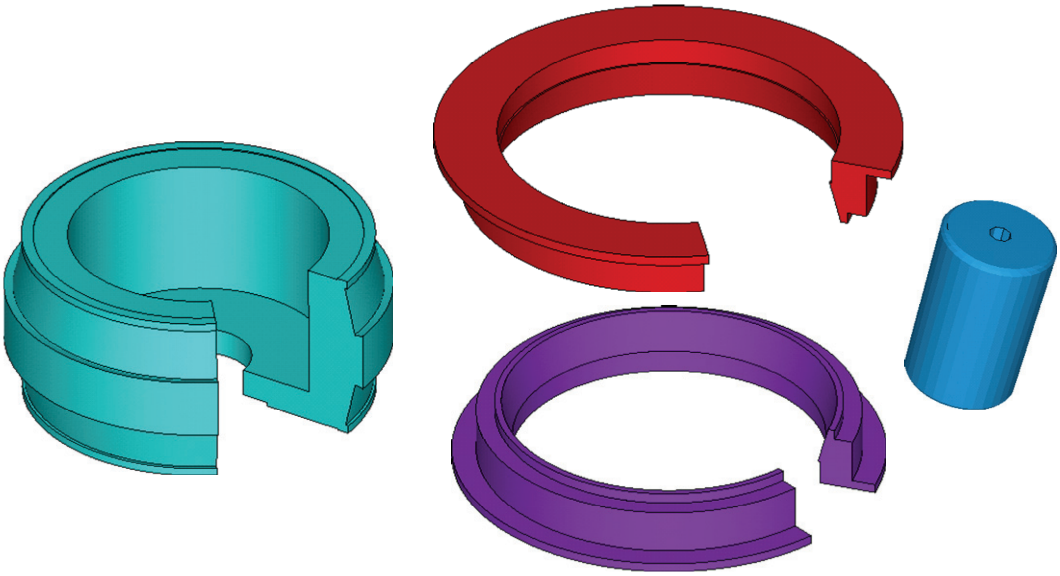

Fig. 4 The inner ring, both outer rings and the roller

Each roller has the contact with inner and outer ring, it means 140 contact surfaces in total. For such a number of contact pairs it was not possible to use the "contact wizard" in the Ansys program. To generate the 140 meshes of contact elements the macro was written. This macro contains the cycle of 35 loops. In every loop the nodes on the rings and the rollers were selected on the ring circumference by 360/35 degrees and then on the lower and upper ring. On selected nodes the contact (the roller) and target (the ring) elements were generated.

The result of the non-linear static analysis is the distribution of the contact pressure on the contact areas (see Fig. 5).

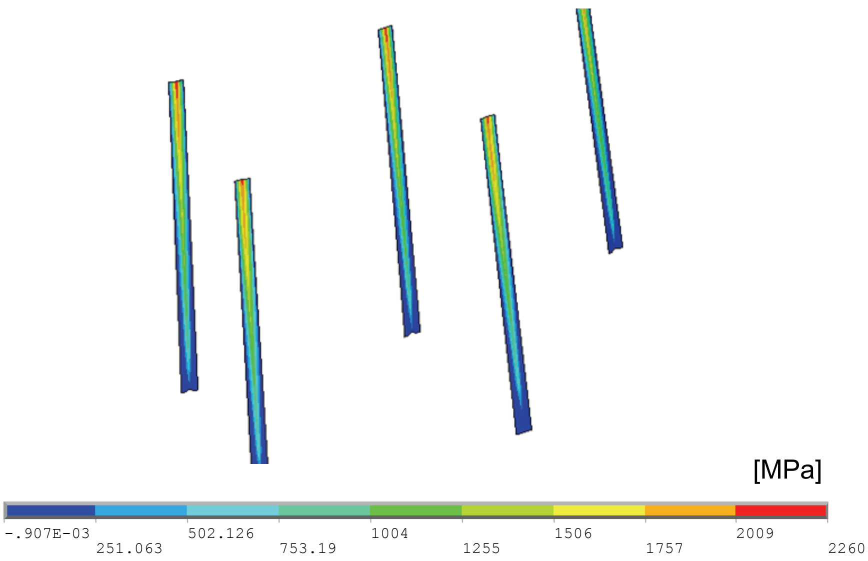

Fig. 5. The distribution of the contact pressure, the detail

5 THE SUPER-ELEMENT MODEL

To build the super-element model it was necessary to define the super-elements. The 73 super-elements were defined - the inner ring, two outer rings and 70 rollers (see Fig. 4). To define the single super-elements (the rings) the Ansys tools were used. The 70 super-elements (the rollers) were defined in the cycle of 35 loops (one lower and one upper roller in each loop). For this the macro was written.

To define the mesh of contact and target elements the 3D mesh of brick elements must exist. This gives the topology on which the mesh of surface elements is generated automatically. But the super-element model does not contain the brick elements and the topology for the surface mesh does not exist. To built the contact pairs the mesh of surface elements (both contact and target) on the standard model was exported into the special file and then imported into the super-element model. Of course it was strongly necessary to conserve the node numbering.

The final model of super-elements and contact elements was completed by boundary condition (the support) and force loading (the same as on the standard model) and then ready for analysis.

6 CONCLUSIONS

While the standard model of brick elements consists of 2 272 617 DOF, the super-element model consists of 65 940 DOF. This means the strong reduction of the model scale. The prize is rather complicated procedure to build the super-element model with contacts. The question is the efficiency of this approach. For single analysis it is much more effective to use the standard model. The super-element model will be effective to perform a number of analyses with iterations.

Acknowledgement

The work was done under support of the MSM 6198910027 project.

REFERENCES

[1] CRISFIELD M.A. *Non-linear Finite Element Analysis of Solids and Structures*. John Wiley & sons, Chichester, 2000. ISBN 0 471 92956 5.

[2] KOLÁŘ V., NĚMEC I., KANICKÝ V. *FEM - Principy a praxe konečných prvků*. Computer Press, Praha, 1997, ISBN 80-7226-021-9.

[3] ZHI HUA ZHONG. *Finite Element Procedures For Contact - Impact Problems*. Oxford University Press, Oxford, 1993, ISBN 0 19 856383 3.

[4] Substructuring. The Ansys manual.

[5] The Non-linear Analysis. The Ansys manual.