17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9 – 12 May 2011

THE DOMAIN DECOMPOSITION WITH THE CONTACT PROBLEM

J. Podešva*

Abstract: *The modeling of the mechanical structures with large number of degrees of freedom (DOF) was in the past the problem of the machine time and disk space. On the today's hardware it is possible to solve 106 equations without big problem. But if the algorithm requires the iteration approach (non-linearities) or solving the dynamic problem via direct numerical integration this could take long time and large disk space. In this case the domain decomposition could give the advantage. The paper describes the solution of the large mechanical structure of the bearing with rather high number of contact pairs using so called "super-elements".*

Keywords: ***Domain decomposition, substructures, super-elements, contact.***

1. Introduction

The domain decomposition is the special approach to solving the large number of equations. It belongs to the group of elimination methods of strong decreasing the number of degrees of freedom (DOF). It consist in selecting the small number of DOF and solving this small system of equations. This takes short time and needs small disk space. The solution is then expanded to the original set of DOF.

The subject of modeling in this paper is the roller bearing with two rings and large number of rolling elements (balls or cylinders). Every touch line between the ring and the roller element represents the contact pair.

The model has four attributes :

– large number of DOF (fine model with large number of small finite elements),
– is naturally divided into substructures (rings and roller elements),
– the single substructures touch the others in the cramped area,
– the touch area between rings and roller elements is a typical contact pair.

Both static and dynamic analysis of the mechanical behavior of such a roller bearing brings the necessity of solution of the large system of equations with non-linear problem - contact.

For such problems the special solvers were developed (Daněk, 2003, Dobiáš, 2010).

2. The Domain Decomposition

The domain decomposition technique allows to strongly decrease the number of DOF. The methods of reduction can be split into two groups.

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

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

The domain decomposition method belongs to the first group.

* assoc. prof.. Ing. Jiří Podešva, Ph.D.: VŠB - Technical University of Ostrava, 17. listopadu 15; 708 33, Ostrava; CZ, e-mail: jiri.podesva@vsb.cz

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

$$\boldsymbol{K} \cdot \boldsymbol{q} = \boldsymbol{f} \tag{1}$$

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 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 :

$$\begin{bmatrix} \boldsymbol{K}_{\text{mm}} & \boldsymbol{K}_{\text{ms}} \\ \boldsymbol{K}_{\text{sm}} & \boldsymbol{K}_{\text{ss}} \end{bmatrix} \cdot \begin{Bmatrix} \boldsymbol{q}_{\text{m}} \\ \boldsymbol{q}_{\text{s}} \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{f}_{\text{m}} \\ \boldsymbol{f}_{\text{s}} \end{Bmatrix} \tag{2}$$

or

$$\begin{aligned} \boldsymbol{K}_{\text{mm}} \cdot \boldsymbol{q}_{\text{m}} + \boldsymbol{K}_{\text{ms}} \cdot \boldsymbol{q}_{\text{s}} &= \boldsymbol{f}_{\text{m}} \\ \boldsymbol{K}_{\text{sm}} \cdot \boldsymbol{q}_{\text{m}} + \boldsymbol{K}_{\text{ss}} \cdot \boldsymbol{q}_{\text{s}} &= \boldsymbol{f}_{\text{s}} \end{aligned} \tag{3}$$

If we will derive from the second group of equations:

$$\boldsymbol{q}_{\text{s}} = \boldsymbol{K}_{\text{ss}}^{-1} \cdot \left(\boldsymbol{f}_{\text{s}} - \boldsymbol{K}_{\text{sm}} \cdot \boldsymbol{q}_{\text{m}} \right) \tag{4}$$

or

$$\boldsymbol{q}_{\text{s}} = \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{f}_{\text{s}} - \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{K}_{\text{sm}} \cdot \boldsymbol{q}_{\text{m}} \tag{5}$$

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

$$\left(\boldsymbol{K}_{\text{mm}} - \boldsymbol{K}_{\text{ms}} \cdot \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{K}_{\text{sm}} \right) \cdot \boldsymbol{q}_{\text{m}} = \boldsymbol{f}_{\text{m}} - \boldsymbol{K}_{\text{ms}} \cdot \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{f}_{\text{s}} \tag{6}$$

After substitution:

$$\begin{aligned} \tilde{\boldsymbol{K}} &= \boldsymbol{K}_{\text{mm}} - \boldsymbol{K}_{\text{ms}} \cdot \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{K}_{\text{sm}} \\ \tilde{\boldsymbol{f}} &= \boldsymbol{f}_{\text{m}} - \boldsymbol{K}_{\text{ms}} \cdot \boldsymbol{K}_{\text{ss}}^{-1} \cdot \boldsymbol{f}_{\text{s}} \end{aligned} \tag{7}$$

the equations have the same form as the original equations.

$$\tilde{\boldsymbol{K}} \cdot \boldsymbol{q}_{\text{m}} = \tilde{\boldsymbol{f}} \tag{8}$$

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

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

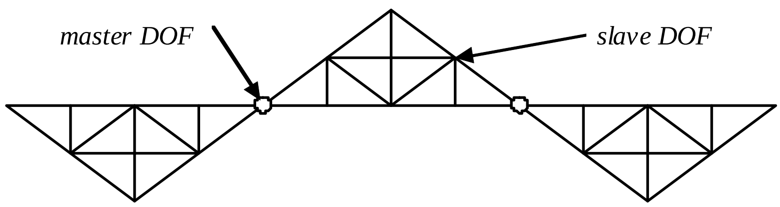

Fig. 1: The main structure divided into three sub-structures.

If the mechanical structure (see Fig. 1) can be naturally dividing into a few sub-structures, these will be the sub-domains. The sub-structures are joined together in the narrow boundaries of the very small number of DOF. These interface DOF will be retained as masters, the interior DOF will be hidden as slaves.

The reduced stiffness matrix $\tilde{\boldsymbol{K}}$ 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”.

The sub-domains (super-elements) must be internally linear. If the super-elements are used to build the “macro model”, this can contain also elements of other types, including contact elements, and other non-linearities.

3. The Contact Problem

The two bodies, contacting one the other, represents the contact problem. It is usually solved by two sets of surface elements, covering the bodies. In the Ansys program they are called “contact elements” and “target elements” (see Fig. 2). One body is covered by contact elements, the other by target elements. The penetration of the contact surface nodes into the target surface is checked during the solution. There are two main methods, the Lagrange method and the penalty method, and the number of auxiliary tools for this.

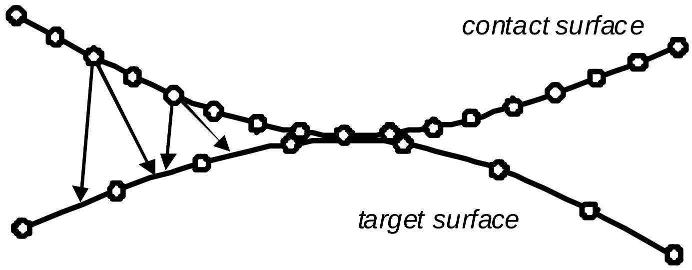

Fig. 2: The contact and the target surface.

The basic principals of mathematical description and solution methods are described in (Crisfield, 2000) and (Zhi Hua Zhong, 1993)

4. The Bearing Model

The modeling of the two row roller bearing (see Fig 3) is described in the paper. The objective is to determine the contact pressure on the rollers. 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).

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.

5. The Super-element Model

The bearing is the typical structure, mentioned in the chapter 2. The natural sub-domains are the inner ring, the two outer rings and the 70 rollers (see Fig. 3). 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). To do this the macro was written.

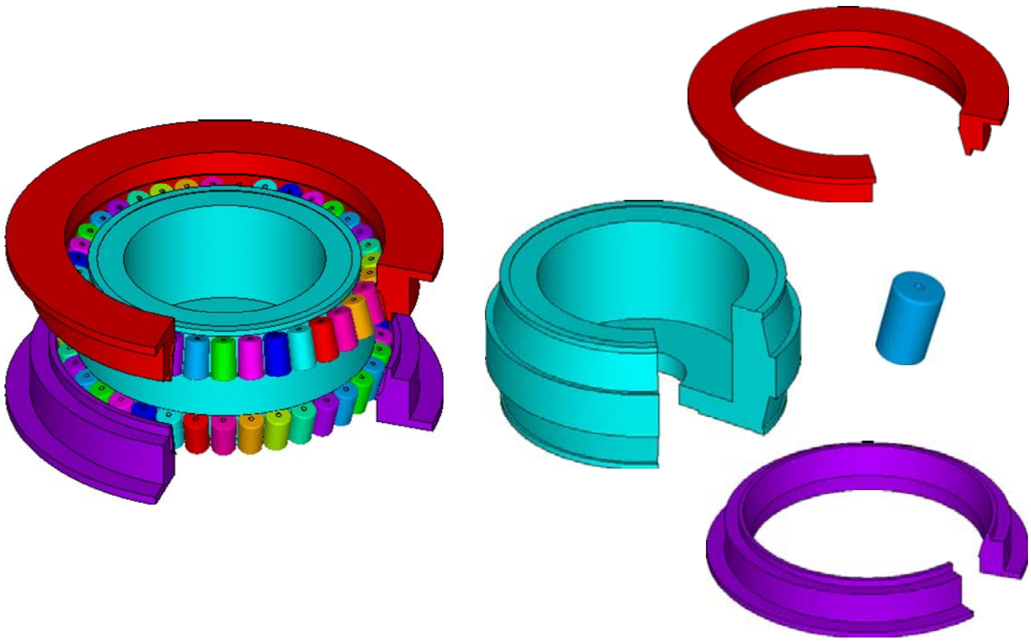

Fig. 3: The bearing and the super-elements.

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. Conclusion

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

Crisfield, M.A. (2000) Non-linear Finite Element Analysis of Solids and Structures. John Wiley & sons, Chichester, 2000. ISBN 0 471 92956 5

Zhi Hua Zhong. (1993) Finite Element Procedures For Contact - Impact Problems. Oxford University Press, Oxford, 1993, ISBN 0 19 856383 3

Daněk, J. (2003) Domain decomposition method for contact problems with small range contact. Mathematics and Computers in Simulation. 2003. ISSN 0378-4754

Dobiáš, J., Pták, S., Dostál, Z., Vondrák, V., Kozubek, T. (2010) Non-linear scalable TFETI domain decomposition based contact algorithm. IOP Conference Series: Materials Science and Engineering. Vol. 10, N. 1. 2010.