

PAPER • OPEN ACCESS

Numerical simulation in CAE Fidesys of bonded contact problems on non-conformal meshes

To cite this article: A V Kukushkin *et al* 2019 *J. Phys.: Conf. Ser.* **1158** 032022

View the [article online](#) for updates and enhancements.

You may also like

- [Anti-resonant reflecting guidance mechanism in SMF-Capillary structure for gas pressure sensing](#)
Y. B. Yang, D. N. Wang, Ben Xu et al.
- [Research on Practical Byzantine Fault Tolerant Consensus Algorithm Based on Blockchain](#)
Xiandong Zheng and Wenlong Feng
- [Structural design of zoomorphic robot](#)
D Durga Prasad and CH Vasanth Kumar

Numerical simulation in CAE Fidesys of bonded contact problems on non-conformal meshes

A V Kukushkin¹, D A Konovalov², A V Vershinin³ and V A Levin³

¹Tula State Pedagogical University, Prospect Lenina 125, Tula, 300026, Russia

²LLC "Fidesys", office 355, building 1, Bolshoy Boulevard 42, Moscow, 121205, Russia

³Moscow State University, GSP-1, Leninskie gory d. 1, Moscow, 119991, Russia

E-mail: versh1984@mail.ru

Abstract. An approach to the numerical simulation of an elastic interaction between the deformable solid bodies inside an assembly is considered. The solids are interacting with each other without sliding and detachment along internal boundaries i.e. solids are stick to each other. An algorithm for the precise modeling of bonded contact interaction between solids is described. Non-conformal spectral element meshes of different orders are used for the discretization of solids. Test examples are presented for the verification of the developed algorithm by comparing obtained numerical results with solutions of similar problems in case of merged solids with a conformal mesh discretization: static and modal analysis of assemblies consisting of cubical and cylindrical bodies. An algorithm's reliability and a continuity of the obtained fields are analyzed in case of gaps/overlaps between contacting solids.

1. Introduction

An approach to the numerical simulation of a contact interaction between the deformable solids inside the assembly is considered in the article. Solids interact with each other during the deformation process without sliding and detachment along internal boundaries (in other words solids are merged to each other). A standard approach for solving such kind of problems is to imprint the boundaries and merge the solids along the common boundary zones. However this approach requires conformal discretization of the whole assembly including conformal meshes on the common boundary regions during the numerical simulation, which often causes significant problems for the industrial assemblies consisting of a large number of parts of different sizes. As a result a mesh could contain a large number of elements, also it is not possible to make a sharp transition from the coarse mesh to the detailed one, to connect meshes with different element types (tetrahedrons, hexahedrons), to generate an unstructured hexahedral mesh in the overall assembly. Moreover, it is difficult or impossible to build a conformal mesh in the overall assembly in case of imperfections (gaps, overlaps, etc. between solids) in the initial geometrical model (which often happens while importing CAD models into CAE systems), and if solids are not ideally attached to each other. As a result it is necessary to heal/modify an initial CAD-model (which is time consuming and not a straightforward process) to build the mesh of acceptable quality.

One of the approaches for solving the described problems with generating meshes for assemblies is to remove a mesh conformity requirement between solids and to build instead independent



discretization in each solid with further tying in order to provide a continuous solution of the boundary-value elasticity problem (stress-strain state parameters) along the boundaries between the solids. A tying algorithm based on the bonded contact interaction between solids is described in the article. A bonded contact between the boundary elements inside the contact region is ensured by direct imposing displacement continuity conditions in the stiffness matrix (and a mass matrix, in case of transient problems) obtained from the discretization of a boundary-value elasticity problem inside the assembly. This is a direct generalization of an approach for setting Dirichlet conditions on displacements in the finite element method. Normal stresses continuity in the contact regions is provided by the corresponding additional terms to the stiffness matrix from boundary integrals along the contact regions as a result of the Galerkin weak formulation (normal stresses continuity in a weak sense). High order space discretization is provided by the spectral element method. A described algorithm allows to obtain a numerical solution for the unstructured non-conformal spectral element meshes (using different spectral element orders in solids), and to provide a continuity in C-norm for primary variables (displacements) and a continuity in L2-norm for normal stresses in the contact region.

Test examples are considered for the verification of the developed algorithm of tying elastic solids. A robustness of the algorithm and a continuity of the obtained solution are analyzed in case of gaps/overlaps between contacting solids. An example of an industrial problem of modal analysis of the micro sputnik assembly is considered.

2. Contact models

In order to find a contact problem solution we modify Finite Element Analysis (FEA) equations by some constraints preventing interacting bodies from interpenetration and providing consistent fields inside them. These constraints are applied to the boundary where interaction occurs. Thus for the boundary AB it is necessary to provide the following boundary conditions [1]:

$$x^1|_{AB} = x^2|_{AB} \text{ and } t^1|_{AB} = t^2|_{AB}, \quad (1)$$

where x^i is deformed position and t^i is the traction between the regions.

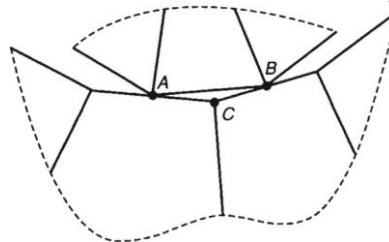


Figure 1. Contact regions with node-face contact [1]

The well-known approaches for introducing these constraints into the global FEA system are:

- Lagrange multipliers method;
- Penalty method;
- Augmented Lagrange multipliers method;
- Constraint elimination method;
- Interior penalty discontinuous Galerkin method [2,3].

All these methods have pros and cons to use them in industrial software solutions. First three methods are the most commonly used so we describe them briefly. Constraint elimination method is the basis for the algorithm considered in the article.

2.1 Lagrange multipliers method

We introduce the Lagrange multiplier functional [4]:

$$\Pi_i = \int_{\Gamma_i} \lambda^T (x^1 - x^2) d\Gamma^i, \quad (2)$$

where $\lambda = t^1 = -t^2$ is the multiplier.

The method introduces new unknown for each contact pair. Also, as for any Lagrange multiplier approach, it gives a zero diagonal element in a global stiffness matrix for each multiplier term. Thus, special care is needed in the solution process to avoid division by the zero diagonal as well as a proper choice of a set of basis functions for an approximation of Lagrange multiplier [5].

2.2 Penalty method

Penalty method uses the following contact term [6] :

$$\Pi_i = \frac{k}{2} \int_{\Gamma_i} (x^1 - x^2)^2 d\Gamma^i, \quad (3)$$

where k is penalty coefficient.

The choice of a penalty coefficient is not an obvious task. If it is too big it leads to an ill conditioned stiffness matrix and if it is too small it leads to large interpenetrations of interacting solids. Traction continuity is not achieved without additional terms.

2.3 Augmented Lagrange multipliers method

Augmented Lagrange multipliers method combines simplicity of the penalty method and also resolves some of its problems with accuracy. The contact potential term has the following form [7]:

$$\Pi_i = \frac{k}{2} \int_{\Gamma_i} (x^1 - x^2)^2 d\Gamma^i + \int_{\Gamma_i} \lambda^T (x^1 - x^2) d\Gamma^i, \quad (4)$$

with λ_i updated on iterations by $\lambda_i^{(m+1)} = \lambda_i^{(m)} - kg^{(m)}$, where $g = (x^1 - x^2)$.

The choice of a penalty coefficient is still a problem, but additional Lagrange multiplier increases accuracy if a penalty coefficient is not large enough. The price for this flexibility is additional iterations which can diverge. Traction continuity in the contact boundary is obtained from the formulation essentially.

2.4 Constraint elimination method

Despite to Lagrange multiplier method constraint elimination method reduces a number of unknowns. It is necessary to define however slave and master degrees of freedom for the constraint system. The global system of FEA equations can be reduced further using these constraints. The selection algorithm of master and slave degrees of freedom is not trivial. In order to impose constraints on displacements using a direct elimination procedure it is necessary first to represent them in the following way [8]:

$$x_s = F(x_m) = \sum_{i=0}^n N(\xi) x_m. \quad (5)$$

These equations are introduced into the global stiffness matrix and global force vector assembly process using one of the elimination techniques [9]. Traction continuity in the contact region needs to be taken into account additionally and will be considered later.

3. Tied contact algorithm

The algorithm may be separated into two steps. First step is a contact pair detection. Second step is an assembly of contact constraints equations and global matrix reduction based on them.

3.1 Contact pair detection

In general case it is quite difficult to set up contact pairs by hand so special algorithm is needed. To minimize user efforts we assume that all surfaces can be in contact and then remove node-face contact pairs which violate the following conditions:

- a node and a face belong to different bodies to prevent self-contact;
- a distance between a face's center and a node is less than some prescribed distance depending on element sizes;
- a distance from a node to its projection on a face is less than a given gap;
- a node's projection on a face's plane is located inside a face;
- an angle between a master surface normal at the projection point and a slave surface normal at a node is larger than a given reference angle (180 degrees in an ideal case).

3.2 Projection point search

Slave node's projection on the master surface can be obtained using the closest point projection method [1]:

$$c = \frac{1}{2} \left(x_k^s - \sum_i N_i(\xi) x_i^m \right)^T \left(x_k^s - \sum_i N_j(\xi) x_j^m \right) = \min. \quad (6)$$

For an interface of a three-dimensional problem it gives the following equations:

$$\frac{\partial c}{\partial \xi_1} = \left(\sum_j \frac{N_j}{\xi_1} x_j^m \right)^T \left(x_k^s - \sum_i N_i(\xi) x_i^m \right) = 0, \quad \frac{\partial c}{\partial \xi_2} = \left(\sum_j \frac{N_j}{\xi_2} x_j^m \right)^T \left(x_k^s - \sum_i N_i(\xi) x_i^m \right) = 0, \quad (7)$$

where $\xi = (\xi_1, \xi_2)$ coordinates of the projection in a reference system of coordinates of an element's face, $N_i(\xi)$ element's face's form functions. This system of equations is nonlinear in general and may be solved only approximately using Newton method [10].

3.3 Constraint matrix

After contact pair detection is done the constraint equation for the pair can be written in the following way using slave node's projection onto the master surface [11]:

$$u_s - \sum_j N_j(\xi) u_j^m = 0, \quad (8)$$

where u_s - slave node displacements, u_j^m - node displacements in a master element, $N_j(\xi)$ - form functions of a master element, ξ - slave node's projection onto master element.

The rectangular system of constraint equations consists of these equations (8) for all slave nodes. Since some nodes could become both master and slave in constraint equations we have to eliminate repeated constraints in order to avoid overconstrained system of equations. So the resulted rectangular system of constraint equations could be transformed into reduced row echelon form (RREF) using linear algebra methods for the rectangular sparse matrix diagonalization (for example, using QR factorization). As a result slave degrees of freedom are separated from master degrees of freedom and constraints could be used for further elimination process at the stiffness matrix assembly stage.

3.4 Traction conditions

Traction continuity conditions on contact boundaries could be imposed in the following way [1, 12]:

$$\int_{\Gamma^s} N_i^s \sigma^m n^s d\Gamma^s + \int_{\Gamma^m} N_i^m \sigma^s n^m d\Gamma^m = 0, \quad (9)$$

where Γ^s - slave surface contact boundary, N_i^s - form functions of a slave element, n^s - normal at a slave surface, σ^m - stress tensor in a master element, Γ^m - master surface contact boundary, N_i^m -

form functions of a master element, n^m - normal at a master surface, σ^s - stress tensor in a slave element.

In general case these additional equations lead to an asymmetric stiffness matrix and as a result general sparse solvers with a higher computational cost should be used for its resolution. However for the case of linear elastic problems with tied contact conditions considered above it is more common to have symmetric stiffness matrix and there are some techniques which could be applied for symmetrizing the matrix [13].

4. Computational results

Both verification and industrial examples are presented in the following section in order to demonstrate an efficiency of the developed approach for handling assemblies with geometrical inaccuracies.

4.1 Linear statics analysis: a cube assembly

As an example of a simple problem demonstrating the continuous displacements and stresses over an assembly a linear elastic problem of a cube assembly bending is presented. Beam is splitted into two parts and gap/intersection is imposed between bodies in order to verify an algorithm tolerance. As it is shown in figures 2 and 3 both displacements and stresses are continuous over non-conformal spectral element meshes with different element orders.

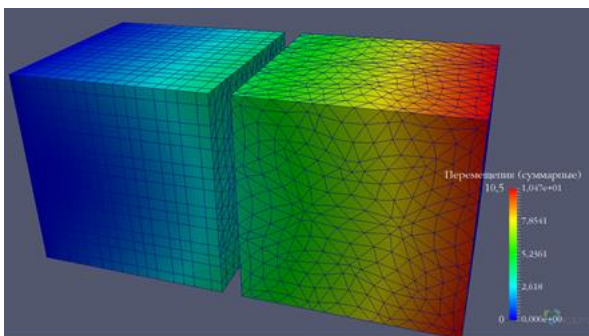


Figure 2. Continuous displacement field over hybrid spectral element meshes of the 3rd (left cube) and 4th (right cube) orders.

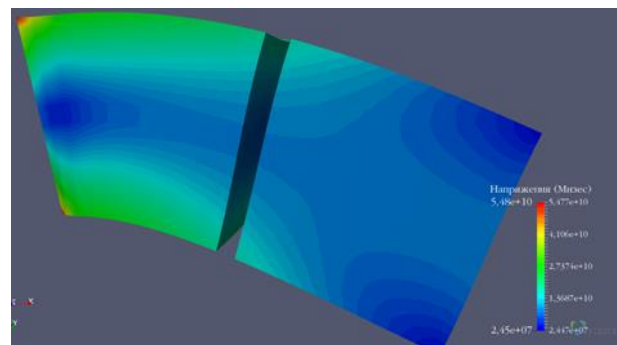


Figure 3. Continuous von Mises stress field.

4.2 Modal analysis: a cube assembly

An example proves algorithm efficiency in a modal analysis of the cube assembly with gaps. As it is shown in figures 4 and 5 stresses are continuous over non-conformal meshes with a gap between bodies.

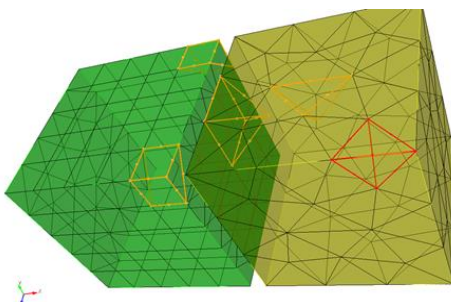


Figure 4. Hybrid finite element mesh for a two-cube assembly.

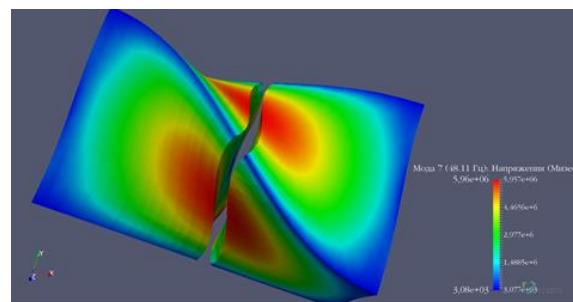


Figure 5. Continuous von Mises stress field for Mode 7.1

4.3 Modal analysis: a cylindrical assembly

An example with two overlapped cylinders is a representative model to demonstrate continuous solution fields at the contact boundary. Figure 6 and 7 shows Mises stress for modes 7 and 9.

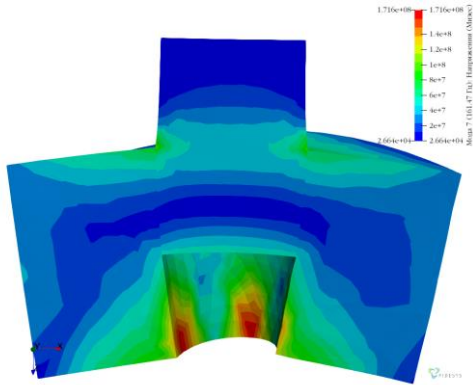


Figure 6. Continuous mode von Mises stress field for Mode 7.

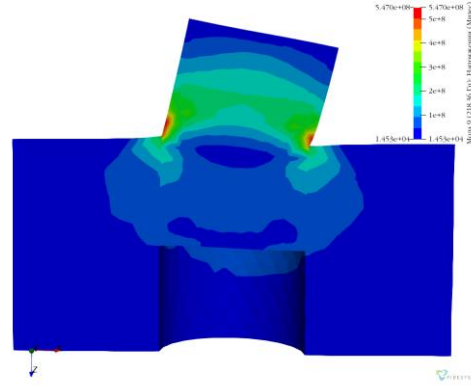


Figure 7. Continuous mode von Mises stress field for Mode 9.

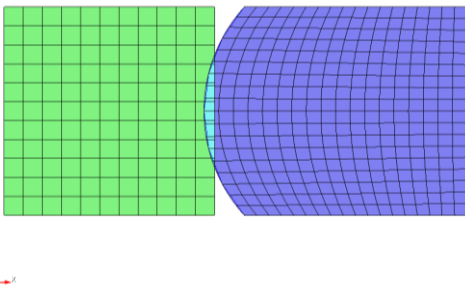


Figure 8. Non-conformal finite element mesh.

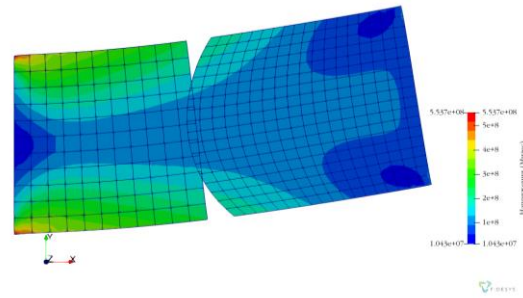


Figure 9. Continuous von Mises stress field on curved contact boundary with gaps and overlaps.

4.4 Linear statics analysis: splitted beam bending with curved boundary

An example with curved boundary inside a splitted beam in figure 9 demonstrates continuous stresses in the contact region with simultaneous overlaps and gaps.

4.5 Industrial examples

The following industrial example of the micro sputnik assembly demonstrates the robustness of the developed automatic contact pair detection algorithm for an overall contact case in figure 10 and 11, continuous stress field in figure 12 and 13. In case of unfixed model first 6 eigenvalues are very close to zeroes, that corresponds to theoretical expectations about 6 possible dimensions of freedom for 3D models. Number of zero eigenvalues is one more criteria to verify the algorithms for bonded contacts.

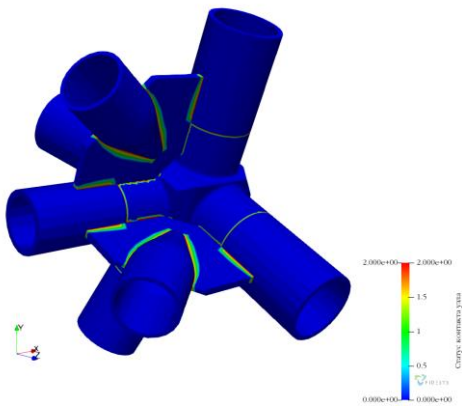


Figure 10. Slave nodes contact status (Red color - nodes are in contact).

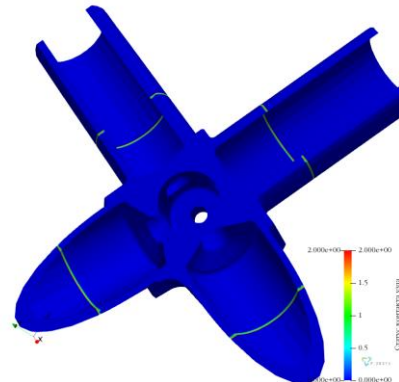


Figure 11. Slave nodes contact status (Red color - nodes are in contact). Section.

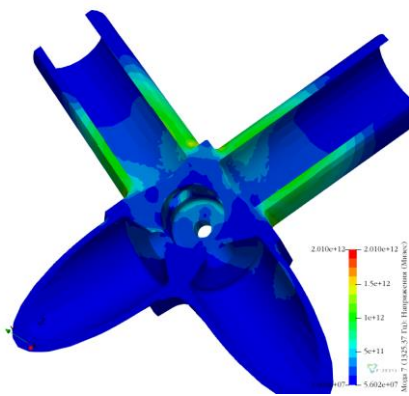


Figure 12. Continuous mode von Mises stress field for Mode 7. Section.

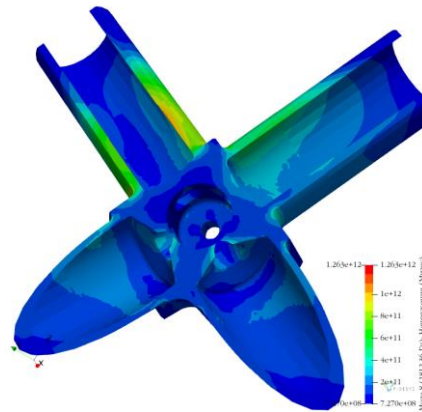


Figure 13. Continuous mode von Mises stress field for Mode 8. Section

5. Conclusion

The described algorithm is implemented in engineering simulation software CAE Fidesys in combination with Finite Element Method (FEM) [1] and Spectral element method (SEM) [14]. Test examples verified the developed algorithm to resolve bonded contact on non-conformal meshes between solids and showed algorithm robustness and solution fields continuity in case of gaps/overlaps between contacting solid bodies. It is shown that small gaps and overlaps in CAD-model of an assembly do not influence much a correctness of numerical results and these cases are correctly and automatically processed in CAE Fidesys software module based on the developed algorithm.

Acknowledgments

The work was performed by the L.N. Tolstoy Tula State Pedagogical University with financial support from the Ministry of Education and Science of the Russian Federation (Project 14.577.21.0271, project ID RFMEF157717X0271).

References

- [1] Zienkiewicz O C and Taylor R L 2014 *The Finite Element Method for Solid and Structural Mechanics, Seventh Edition* (Amsterdam: Elsevier)
- [2] Cockburn B, Karniadakis G E and Shu Chi-Wang 2000 *Discontinuous Galerkin Methods: Theory, Computation and Applications* (Berlin: Springer-Verlag)

- [3] Arnold D N, Brezzi F, Cockburn B and Marini D 2002 Unified analysis of discontinuous Galerkin methods for elliptic problems *SIAM J. Numer. Anal.* **39** 1749–1779
- [4] Papadopoulos P, Solberg J M and Lagrange A 1998 Multiplier method for the finite element solution of frictionless contact problems *Math. Comput. Model.* **28** 373–384
- [5] Popp A, Wohlmuth B, Gee M and Wall W 2012 Dual Quadratic Mortar Finite Element Methods for 3D Finite Deformation Contact *SIAM Journal on Scientific Computing* **34** DOI:10.1137/110848190.
- [6] Shephard M S 1984 Linear multipoint constraints applied via transformation as part of a direct stiffness assembly process *Int. J. Numer. Meth. Engng* **20** 2107-2112
- [7] Abel J F and Shephard M S 1979 An algorithm for multipoint constraints in finite element analysis *Int. J. Numer. Meth. Engng* **14** 464–467
- [8] Landers J A and Taylor R L 1985 An augmented Lagrangian formulation for the finite element solution of contact problems *Technical Report SESM 85/09 University of California, Berkeley*
- [9] Peric D and Owen R J 1992 Computational model for 3-d contact problems with friction based on the penalty method *Int. J. Numer. Meth. Engng* **35** 1289-1309
- [10] Simo J C and Vu-Quoc L 1986 A three-dimensional finite strain rod model. Part II: Geometric and computational aspects *Comput. Methods Appl. Mech. Eng.* **58** 79–116
- [11] Chow E, Manteuffel T A, Tong C and Wallin B K 2001 Algebraic elimination of slide surface constraints in implicit structural analysis *Int. J. Numer. Meth. Engng* **01** 1-21
- [12] Wang X, Prakash A, Chen JS and Tacioglu E 2017 Variationally consistent coupling of non-matching discretizations for large deformation problems *Comput Mech* **60** 465 DOI:10.1007/s00466-017-1417-0
- [13] Felippa C A 2004 *Introduction to finite element methods. Chapter 8 Multifreedom constraints I* (Colorado: Department of Aerospace Engineering Sciences and Center for Aerospace Structures University of Colorado Boulder) 1–17
- [14] Konovalov D A, Vershinin A V, Zingerman K M and Levin V A 2017 The Implementation of Spectral Element Method in a CAE System for the Solution of Elasticity Problems on Hybrid Curvilinear Meshes *Modelling and Simulation in Engineering* DOI:10.1155/2017/1797561