Structural analysis of assemblies using non-conformal spectral element method

A V Vershinin¹, V A Levin¹, A V Kukushkin², D A Konovalov²
¹ Lomonosov Moscow State University
² Fidesys LLC
versh1984@mail.ru

Abstract. An approach to the numerical simulation of a contact interaction between the deformable solids inside the assembly is considered in the article. 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. Test examples are considered for the verification of the developed in CAE Fidesys (www.cae-fidesys.com) algorithm of tying elastic solids by comparing simulation results with the solutions of similar problems for the case of merged solids with a conformal mesh discretization: static and modal analysis of the assemblies consisting of cubic, cylindrical and spherical bodies. A robustness of the algorithm and a continuity of the obtained solution are analyzed in case of gaps/overlaps between contacting solids. 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 described algorithm.

1. Introduction

The article considers a structural analysis of assemblies (consisting of several solids) using non-conformal discretization by spectral element method. 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). Conformal discretization of the problem results in a mesh containing 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 stress (traction) 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 $C^1$-norm for primary variables (displacements) and a continuity in $L^2$-norm for normal stresses in the contact region.

Test examples are considered for the verification of the developed in CAE Fidesys (www.cae-fidesys.com) algorithm of tying elastic solids by comparing simulation results with the solutions of similar problems for the case of merged solids with a conformal mesh discretization: static and modal analysis of the assemblies consisting of cubic, cylindrical and spherical bodies. A robustness of the algorithm and a continuity of the obtained solution are analyzed in case of gaps/overlaps between contacting solids. 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 described algorithm. An example of an industrial problem of modal analysis of the micro sputnik composite part is considered.

2. Problem statement
The mathematical model of the considered problem consists of the linear momentum equations (Zienkiewicz, 2014):

$$\nabla \cdot \sigma + P = \rho \ddot{u}$$

(1)

$\sigma$ - stress tensor, $P$ - volumetric forces, $\rho$ – density, $\ddot{u}$ - acceleration.

Boundary conditions

$$u|_{\Gamma_u} = \bar{u}$$

(2)

$$\bar{t}|_{\Gamma_t} = \bar{\tau}$$

(3)

Here $\Gamma_u$ - is a part of the boundary where displacement specified, $\Gamma_t$ is a part of the boundary where external traction applied.

Constitutive relations for Hooke’s material (Zienkiewicz, 2014):

$$\sigma = D\varepsilon$$

(4)

For three-dimensional case:

$$\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\sigma_{23} \\
\sigma_{13} \\
\sigma_{12}
\end{bmatrix} =
\begin{bmatrix}
1 - \nu & \nu & \nu & 0 & 0 & 0 \\
\nu & 1 - \nu & \nu & 0 & 0 & 0 \\
\nu & \nu & 1 - \nu & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
\varepsilon_{23} \\
\varepsilon_{13} \\
\varepsilon_{12}
\end{bmatrix}$$

(5)

Here $E$ – Young modulus, $\nu$ – Poisson coefficient.

3. Contact constraints
Contact conditions (Zienkiewicz, 2014) for tied surfaces include a continuity of displacements:

$$x^1|_{AB} = x^2|_{AB}$$

(6)

And a continuity of the traction:
Traction continuity is not achieved in the case of non-conformal meshes without a special care.

\[ t^1_{\Gamma B} = -t^2_{\Gamma B} \]

Here \( x^i \) are coordinates in the current (deformed) state and \( t^i \) is the traction at the interface between the tied solids.

4. **Numerical model**

Presented above partial differential equations can be reformulated in an integral (weak) form using Galerkin method (Zienkiewicz, 2014):

\[
\int_{\Omega} \nu \cdot \rho \ddot{u} d\Omega + \int_{\Omega} \nabla \nu \cdot \sigma d\Omega = \int_{\Gamma} \nu \cdot P d\Gamma, \quad \forall \nu \in H^1.
\]

Spectral element method (SEM) (Komatitsch, 1998) is used for the discretization of the integral equations as it has some advantages over classical finite element method (FEM) (Zienkiewicz, 2014): high accuracy of the numerical solution compared with FEM, efficient parallelization thanks to the locality property i.e. each spectral element could be solved independently in computational domains.

Let us assume that the number of nodes in a spectral element is \( m \), then the number of basis functions \( (N_k \in H^1) \) is \( m \) too, and as a result an approximate solution \( u^e = u|_{\Omega^e} \) in an element \( \Omega^e \) could be represented in the following way: \( u^e \approx \sum_{k=1}^{m} N_k U^e_k \). Substituting it to the Galerkin equations and using spectral element discretization of the domain \( \Omega \) one obtains:

\[
\sum_{e=1}^{n_e} \int_{\Omega^e} \nu \cdot \rho \ddot{u^e} d\Omega - \sum_{e=1}^{n_e} \int_{\Omega^e} \nabla N_k \cdot \sigma d\Omega = \sum_{e=1}^{n_e} \int_{\Omega^e} N_k \cdot P d\Omega, \quad k = 1..m.
\]

Let us define \( \mathbf{N} = (N_1, \ldots, N_m) \), \( (\mathbf{U}^e)^T = (U^e_1, \ldots, U^e_m) \), then a system of integral equations can be rewritten in the following way:

\[
\sum_{e=1}^{n_e} \mathbf{M}^e \ddot{\mathbf{U}}^e - \sum_{e=1}^{n_e} \mathbf{K}^e (\mathbf{U}^e) - \sum_{e=1}^{n_e} \mathbf{P}^e = 0, \quad k = 1..m.
\]

\[
\mathbf{M}^e = \int_{\Omega^e} \mathbf{N}^T \rho \mathbf{N} d\Omega - \text{local mass matrix of an element } \Omega^e,
\]

\[
\mathbf{K}^e (\mathbf{U}^e) = \int_{\Omega^e} \mathbf{N}^T \cdot \mathbf{\nabla} \sigma d\Omega - \text{internal forces vector of an element } \Omega^e,
\]

\[
\mathbf{P}^e = \int_{\Gamma^e} \mathbf{\nabla} \cdot \mathbf{P} d\Gamma - \text{external forces vector of an element } \Omega^e.
\]

5. **Contact constraints implementation**

In order to satisfy contact constraints presented above we will introduce them into the system of ODE obtained in the previous paragraph. There are several approaches for introducing these constraints into the global FEA system:

- Penalty method (Peric 1992);
- Lagrange multipliers method (Papadopoulos 1998);
- Constraint elimination method (Multi point constraints, MPC) (Shephard 1984);
- Interior penalty discontinuous Galerkin method (Cockburn 2000, Arnold 2002);

Each of them has some advantages and disadvantages. Let us look at them in details.

5.1. **Penalty method**

Penalty method uses the following contact term (Peric 1992) in the potential energy representation:

\[
\Pi_i = \frac{k}{2} \int_{\Gamma_i} (x^1 - x^2)^2 d\Gamma_i
\]

Here \( k \) is a 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 in the case of non-conformal meshes without a special care.

5.2. **Lagrange multipliers method**

The Lagrange multiplier functional has the following form (Papadopoulos 1998):

\[
\Pi_i = \int_{\Gamma_i} \lambda (x^1 - x^2) d\Gamma_i
\]
Here $\lambda = t^4 = -t^2$ is the Lagrange multiplier introduced at the contact boundary.

The method introduces a 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, a special kind of iterative sparse solver (Uzawa 1958) is needed in order to avoid division by zero as well as a proper choice of basis functions for an approximation of Lagrange multiplier (Landers 1985) for example by mortar or dual-mortar methods (Popp 2012).

5.3. Constraint elimination method

Constraint elimination method reduces a number of unknowns in a global system of FEA equations. However, it requires a proper choice of slave and master degrees of freedom for the constraint system. The global system of FEA equations can be reduced further using these constraints. The algorithm of assigning 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 of all to represent all constraints in the following way:

$$x_s = F(x_m) = \sum_{i=0}^{n} N_i(\xi)x_m$$

here $x_s$ is slave node coordinate, $x_m$ master node coordinate, $N$ – basis function, $\xi$ – local coordinates of the slave node projection at the master surface.

These equations are introduced further into the global stiffness matrix and global force vector at a standard assembly process using one of the elimination techniques (Shephard 1984).

Traction continuity conditions on contact boundaries could be imposed in the following way:

$$\int_{\Gamma_s} N_i^s \sigma^{m}s d\Gamma^s + \int_{\Gamma_m} N_i^m \sigma^s n^m d\Gamma^m = 0$$

here $\Gamma_s$ – slave surface contact boundary, $N_i^s$ - basis functions of a slave element, $n^s$ - a normal to the slave surface, $\sigma^m$ - stress tensor of a master element, $\Gamma^m$ - master surface contact boundary, $N_i^m$ - basis functions of a master element, $n^m$ - a normal to the master surface, $\sigma^s$ - stress tensor of a slave element.

In general case these additional terms in Galerkin weak formulation lead to an asymmetric stiffness matrix and as a result general sparse solvers with a higher computational cost must be used. However for the case of linear elastic problems with tied contact conditions considered above it is common to obtain a symmetric stiffness matrix and there are some techniques which could be applied for symmetrizing the matrix (Felippa 2002).

6. Computational results

Some verification examples and industrial applications of the developed approach are shown below to demonstrate its robustness and efficiency in handling assemblies with geometrical inaccuracies (gaps, overlaps between solids) using high order spectral elements.

6.1 Modal Analysis of a two-cube assembly with a gap

One of the possible applications of the developed algorithm is the modal analysis of assemblies with geometrical inconsistencies in order to obtain continuous displacements and stresses at different modes. An example of such problem is a modal analysis of a two cubes assembly with a gap between them. Both cubes are 1m x 1m x 1m, their material properties: Young’s modulus 211 GPa, Poisson ration 0.3, Density 8000 kgm$^3$. Boundary conditions: tied contact between cubes. A coarse non-conformal mesh was generated for cubes with 4-th and 3-rd orders of spectral elements in the cubes correspondingly. Figures 1 and 2 shows the continuous result fields through the gap between solids.
6.2 Modal analysis: a cylindrical assembly with an overlap
An example with two overlapped cylinders is a representative model to demonstrate continuous solution fields at the contact boundary. Figures 3 and 4 show the non-conformal mesh for the cylinders and von Mises stresses for the 7th eigenmode correspondingly.

6.3 Kirsch problem at non-conformal meshes
The Kirsch problem was solved to demonstrate stress continuity if proposed contact solution method used to refine the mesh close to a stress concentrator. Stress on the boundary is 1e6 Pa. Analytical solution gives maximal stresses at the boundary of the stress concentrator of 3e6 Pa. The stress distribution is presented on figure 5 for the second order curvilinear non-conformal mesh. Results for the conformal mesh are presented on figure 6. Note the similarity of both fields.
6.4 An industrial example
The following industrial example of the tee assembly demonstrates the robustness of the developed automatic contact pair detection algorithm for the whole assembly (figures 7 and 8). Continuous stress fields are presented on figures 9 and 10. In the case of a free model, the first 6 eigenvalues are very close to zero, which is explained by the fact that any free 3D model has 3 translational and 3 rotational DOFs of rigid body motion. Thus a number of zero eigenfrequencies of a free assembly is another criteria to verify the algorithms for bonded contacts. The eigenfrequencies for conformal and non-conformal meshes are compared at Table 1.
Figure 9. Continuous von Mises stresses for the eigenmode 7.

Figure 10. Continuous von Mises stresses for the eigenmode 8.

Table 1. Eigenfrequencies for conformal and non-conformal meshes.

| № of eigenfrequency | Conformal mesh | Non-conformal mesh | Difference % |
|---------------------|---------------|-------------------|--------------|
| 1                   | 0             | 0                 | 0            |
| 2                   | 0             | 0                 | 0            |
| 3                   | 0             | 0                 | 0            |
| 4                   | 0             | 0                 | 0            |
| 5                   | 0             | 0                 | 0            |
| 6                   | 0             | 0                 | 0            |
| 7                   | 1 068.29      | 1 095.18          | 2.52         |
| 8                   | 1 172.06      | 1 142.34          | 2.54         |
| 9                   | 3 445.75      | 3 550.71          | 3.05         |
| 10                  | 3 528.57      | 3 611.89          | 2.36         |

7. Conclusion
An algorithm for the automatic tying the solids along contact boundaries is developed. It allows using non-conformal discretization with spectral elements of different orders in contacting solids. It also correctly handles the cases of initial geometrical inaccuracies (gaps, overlaps) presented inside of the assembly between solids. In all cases continuous displacement and traction fields are obtained at the contact boundaries. The described algorithm is implemented in engineering simulation software CAE Fidesys in combination with Finite Element Method (FEM) (Zienkiewicz, 2014) and Spectral element method (SEM) (Konovalov, 2017). Presented examples of static and modal analysis problems verified the algorithm’s capabilities in resolving bonded contact on non-conformal meshes and demonstrated
an algorithm’s robustness and a continuity of obtained numerical results in case of gaps/overlaps between contacting solids. It is shown that small gaps and overlaps in the initial 3D geometrical representation of an assembly do not influence much obtained numerical results and these inconsistencies are correctly and automatically processed by the suggested algorithm. Thus the developed approach can be used for an engineering analysis of CAD models causing considerable difficulties in their conformal meshing which is one of the major typical roadblocks for non-expertised CAE users, and correspondingly this may support further CAE democratization.

8. References

[1] Abel J and Shephard M 1979 An algorithm for multipoint constraints in finite element analysis *Int. J. Numer. Meth. Engng* 14 pp 464–467
[2] Arnold D, Brezzi F, Cockburn B and Marini D 2002 Unified analysis of discontinuous Galerkin methods for elliptic problems *SIAM J. Numer. Anal.* 39 pp 1749–1779
[3] Charara M, Vershinin A, Deger E, Sabitov D and Pekar G 2011 3D spectral element method simulation of sonic logging in anisotropic viscoelastic media *SEG Expanded Abstracts* 30 pp 432–437
[4] Charara M, Vershinin A, Sabitov D and Pekar G 2011 SEM wave propagation in complex media with tetrahedral to hexahedral mesh *73rd European Association of Geoscientists and Engineers Conference and Exhibition* pp 41–45
[5] Chow E, Manteuffel T, Tong C and Wallin B 2001 Algebraic elimination of slide surface constraints in implicit structural analysis *Int. J. Numer. Meth. Engng* 01 pp. 1-21
[6] Cockburn B, Karniadakis G and Shu Chi-Wang 2000 Discontinuous Galerkin Methods: Theory, Computation and Applications *Springer-Verlag*
[7] Felippa C 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* pp 1-17
[8] Karpenko V, Vershinin A, Levin V and Zingerman K 2016 Some results of mesh convergence estimation for the spectral element method of different orders in Fidesys industrial package *IOP Conference Series: Materials Science and Engineering* 158
[9] Komatitsch D and Tromp J 2002 Spectral-element simulations of global seismic wave propagation-I. Validation *Geophysics Journal International* p 149.
[10] Komatitsch D and Villette J-P 1998 The spectral element method: an efficient tool to simulate the seismic response of 2D and 3D geological structures *Bulletin of Seismological Society of America* 88(2)
[11] Konovalov D, Vershinin A, Zingerman K and Levin V 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*
[12] Landers J and Taylor R 1985 An augmented Lagrangian formulation for the finite element solution of contact problems *Technical Report SESM* 85/09 University of California, Berkeley.
[13] Levin V, Zingerman K, Vershinin A, Freiman E and Yangirova A 2013 Numerical analysis of the stress concentration near holes originating in previously loaded viscoelastic bodies at finite strains *International Journal of Solids and Structures* 50(20-21) pp 3119–3135
[14] Levin V, Zingerman K, Vershinin A and Nikiforov I 2012 CAE Fidesys for strength analysis at large strains and their redistribution *10-th Word Congress on Computational Mechanics* (Sao Paulo: Book of Abstracts) pp 323–323
[15] Papadopoulos P, Solberg J and Lagrange A 1998 Multiplier method for the finite element solution of frictionless contact problems *Math. Comput. Model* 28 pp 373–384
[16] Peric D and Owen R 1992 Computational model for 3-d contact problems with friction based on the penalty method *Int. J. Numer. Meth. Engng* 35 pp 1289-1309
[17] 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

[18] Shephard M 1984 Linear multipoint constraints applied via transformation as part of a direct stiffness assembly process Int. J. Numer. Meth. Engng 20 pp 2107-2112

[19] Simo J and Vu-Quoc L 1986 A three-dimensional finite strain rod model. Part II: Geometric and computational aspects Comput. Methods Appl. Mech. Eng. 58 pp 79–116

[20] Uzawa H 1958 Iterative methods for concave programming Studies in linear and nonlinear programming 6

[21] Wang X, Prakash A, Chen J and Taciroglu E 2017 Variationally consistent coupling of non-matching discretizations for large deformation problems Comput. Mech. 60 p 465

[22] Zienkiewicz O and Taylor R 2014 The Finite Element Method for Solid and Structural Mechanics Seventh Edition (Amsterdam: Elsevier)

[23] www.cae-fidesys.com

Acknowledgments
The reported study was partially funded by Russian Science Foundation project № 19-77-10062 (sections 1-4) and by RFBR and Moscow city Government according to the research project № 19-38-70001 (sections 5-7).