

Development of a Finite Volume Contact Solver Based on the Penalty Method

P.Cardiff^{a,*}, A.Karač^b, A.Ivanković^a

^a*School of Mechanical and Materials Engineering, University College Dublin, Belfield, D4, Dublin, Ireland*

^b*Faculty of Mechanical Engineering, University of Zenica, Fakultetska 1, 72000 Zenica, Bosnia and Herzegovina*

Abstract

This paper describes the development and application of a frictionless contact stress solver based on the cell-centred finite volume method. The contact methodology, implemented in the open-source software OpenFOAM, is derived from the penalty method commonly used in finite element contact algorithms. The solver is verified on two benchmark tests using the available Hertzian analytical solutions.

Keywords: contact stresses, finite volume method, OpenFOAM, iterative penalty method

1. Introduction

The finite element method has dominated numerical structural analysis for the past half century. However, since the early work from Demirdžić et al. [1], finite volume (FV) stress analysis has become a viable alternative for many problems. Nevertheless, there is limited work done in development of finite-volume based contact procedures. Jasak and Weller [2] developed a simple contact solver for steady-state problems, which was later on extended for dynamic problems by Tropsa et al. [3]. These solvers were very effective for 2D problems, but produced some unrealistic stress peaks at the edge of contact areas for 3D problems. To overcome these difficulties, the current

*Corresponding author

Email addresses: philip.cardiff@ucd.ie (P.Cardiff), akarac@mf.unze.ba (A.Karač), alojz.ivankovic@ucd.ie (A.Ivanković)

authors have developed a new correction technique at the contact boundary. The procedure is broadly based on the penalty and augmented Lagrangian methods commonly seen in the finite element contact algorithms [4, 5].

2. Numerical Method

The contact stress solver has been implemented in OpenFOAM-1.6-ext (Open Field Operation And Manipulation) [6]. OpenFOAM is a general 3D based, open source, object-oriented C++ computational continuum mechanics library. In this solver, bodies in contact are assumed linear-elastic with thermal effects being neglected. In addition to the standard field equations [6], contact adds frictional and impenetrability constraints [4]. The current contact solver is frictionless and as such the frictional constraints will not be discussed here. On the other hand, the impenetrability contact constraints state that: (i) $d_N \geq 0$, *i.e.* no penetration may occur; d_N is the normal distance to the opposite contact surface and is negative for a point that crosses over the opposite contact surface, (ii) $t_N \leq 0$, *i.e.* the contact normal traction should be compressive or the contact surfaces cannot withstand tensile forces, and (iii) $t_N \cdot d_N = 0$, the complementarity condition, *i.e.* if there is no contact, then no compressive tractions can occur; alternatively, if there are no compressive stresses, the distance must be positive.

3. Contact Algorithm

At the start of the analysis, the two potentially contacting surfaces are designated as the master and slave surfaces. During the iterative solution, the slave vertices are checked for penetration of the master surface. If a slave vertex does penetrate, an interface force is applied between the slave vertex and the master surface, in the direction of the slave normal with the magnitude proportional to the amount of penetration. The slave vertex forces, \mathbf{F}_{sv} , are calculated as:

$$\mathbf{F}_{sv} = \begin{cases} -p_{sv}k\mathbf{n} & \text{for } p_{sv} < 0 \\ 0 & \text{for } p_{sv} \geq 0 \end{cases} \quad (1)$$

where the slave vertex penetrations, p_{sv} , are calculated as described by Jasak and Weller [2], \mathbf{n} are the slave vertex normals, and k is the interface stiffness,

also known as the penalty factor or penalty stiffness. As an initial guess for the penalty factor, one can use the following equation [5]:

$$k = \frac{f_{scale} K A^2}{V} \quad (2)$$

where K is the bulk modulus of the master, A is the area of the contact master face and V is the volume of the contacting master cell. In finite element algorithms, the scaling factor, f_{scale} , is typically set to 0.1 or less to avoid instability issues. However, it has been found that for our FV-based solver f_{scale} can be set to a substantially larger values and still maintain contact convergence. Calculated slave vertex forces are then interpolated to the slave boundary face centres and applied as the traction boundary condition. If the analysis involves a deformable master body then the slave face tractions are interpolated to the master faces using a strongly conservative General Grid Interface interpolation method [7]. For the rigid master body, however, the the master surface is set to traction free. As the linear momentum equation is iteratively solved for displacement, the contact boundary updating procedure is repeatedly invoked. Corrective contact forces are incrementally added to penetrating slave vertices until they are either no longer in contact or lie within the defined contact tolerance. To avoid instabilities, vertex forces are slowly reduced to zero on the vertices leaving the contact.

The algorithm explained above is parallelised, *i.e.* the contact solver can be used on a distributed-memory parallel machine. To allow the contact boundaries to be corrected in a consistent manner, each processor has a copy of the full contact boundary surfaces through use of global face zones in OpenFOAM. In this way, each processor core communicates with each other processor core during correction of the contact boundaries.

4. Validation Cases

The developed contact solver is used to simulate contact between a deformable hemisphere, radius of 1 mm, and deformable/rigid brick, $2 \times 2 \times 1$ mm, shown in figure 1(a). The problem geometry and mesh, containing 54,628 hexahedral cells, have been created in commercial meshing software ANSYS ICEM CFD. A Young's modulus of 500 MPa, and a Poisson's ratio of 0.3 is employed for all deformable parts. The top surface of the hemisphere is displaced 0.01 mm in the negative y -direction, while the bottom of the brick is fixed. All other surfaces are traction-free. For the contact algorithm, the

curved surface of the hemisphere is designated the slave and the top surface of the brick is the master.

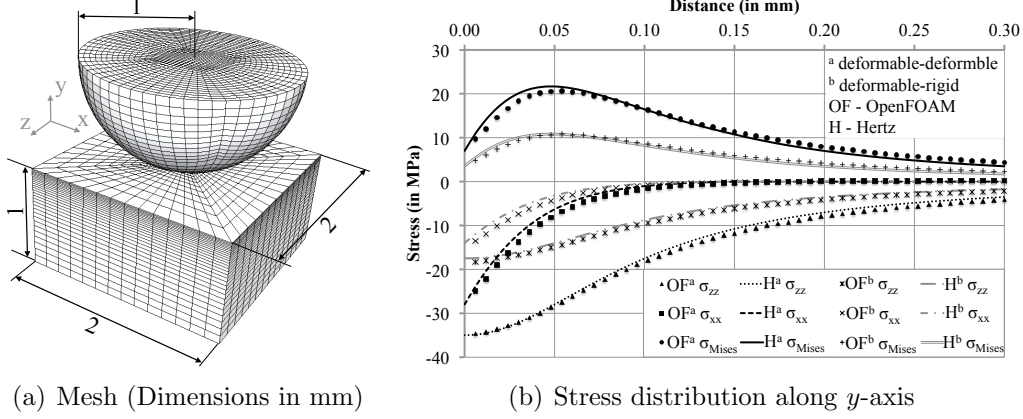


Figure 1: *Deformable Sphere-Deformable/Rigid Brick Test Cases*

A comparison of the stress results for the *deformable sphere-deformable/rigid brick* test cases with the Hertzian analytical solution is shown in figure 1(b). The stresses are plotted from the centre of contact along the y axis of the sphere. OpenFOAM predicts a maximum contact pressure of 18.23 MPa in the deformable brick case and 34.63 MPa in the rigid brick case, compared with the Hertzian predictions of 17.65 MPa and 34.73 MPa respectively. Inspecting contact areas, Hertzian theory predicts a contact radius of 0.1 mm for the *deformable sphere-rigid brick* case, and OpenFOAM gives a contact radius of 0.107 mm, while for the *deformable sphere-deformable brick* case, the Hertzian contact radius is 0.1 mm, and the OpenFOAM contact radius is 0.109 mm.

5. Conclusions

An iterative penalty method contact stress methodology implemented in a cell-centred finite volume framework has been presented. The accuracy of the current procedure has been validated by comparison of *sphere-brick* simulation test cases with the Hertzian analytical solutions. The maximum contact pressures and contact areas show very good agreement, the small discrepancies being due to the Hertzian assumption of the loading force as apposed to the loading displacement.

Although the developed methodology was successful in validation, there are some limitations. Choice of which surface is master or slave still causes problems in some situations, and this is currently investigated. Also, the converged contact faces sometimes display a *zig-zag*. This effect can be reduced by setting a fine contact tolerance, but some other (rectifying) procedures, such as vertex smoothing and corrective shear forces, are presently under investigation.

6. References

- [1] I. Demirdžić, D. Martinović, A. Ivanković. Numerical Simulation of Thermal Deformation in Welded Workpiece (in Croatian), Zavarivanje, 31:209–219, 1988.
- [2] H. Jasak, H. Weller. Finite Volume Methodology for Contact Problems of Linear Elastic Solids. In Proceedings of 3rd International Conference of Croatian Society of Mechanics, Cavtat/Dubrovnik, 253–260, 2000.
- [3] V. Tropsa, I. Georgiou, A. Ivankovic, A. J. Kinloch, J. G. Williams. OpenFOAM in Non-linear Stress Analysis: Modelling of Adhesive Joints. 1st OpenFOAM Workshop, Zagreb, 2006.
- [4] T. Belytschko, W. Kam Liu, B. Moran. Nonlinear Finite Elements for Continua and Structures. Wiley, 2000.
- [5] J. Hallquist, G. Goudreau, D. Benson. Sliding Interfaces with Contact-Impact in Large-Scale Lagrangian Computations. Computer Methods in Applied Mechanics and Engineering, 51:107–137, 1985.
- [6] H. G. Weller, G. Tabor, H. Jasak, C. Fureby. A Tensorial Approach to Computational Continuum Mechanics Using Object Orientated Techniques. Computers in Physics, 12 (6):620–631, 1998.
- [7] M. Beaudoin, H. Jasak. Development of a Generalized Grid Interface for Turbomachinery Simulations with OpenFOAM. OpenSource CFD International Conference, Berlin, 2008.
- [8] K. L. Johnson. Contact Mechanics. Cambridge University Press, 1987.