Fast contact method for speeding up solving finite element problems involving non-linear contact behavior

Numair Mazgaonkar
Senior Application Engineer

Abstract

For large aerospace assemblies in finite element (FE) analysis problems, contact interaction between the surrounding bodies has to be established to simulate the load transferred between the components, like aircraft engine carrying bracket assemblies, spigots assemblies etc., and understand the effects of interaction between respective parts. In some cases, depending upon geometry of the assembly, the region of study may not be contact area but the stresses acting within the parts themselves. If there is no geometric or material non-linearity in such problems, a new contact formulation method known as Fast Contact can be used in these contact regions. In this method, contact non-linearity could be introduced to simulate the problem but friction between the contacting parts should not be present. Currently, there is a scope for applying this method for solving FE problems in the aerospace and rail industry. This paper focuses on this problem-solving method which has been developed by Altair's OptiStruct to solve such FE problems accurately in short period of time. To show the advantages of time savings, this method has been applied to a real aerospace engine bracket model. Theoretical validation of this method is shown by comparing it with the Hertz contact theory. A model of the sphere-section in contact with the rectangular plate is used to prove the validation. Further, this method is also compared against the standard non-linear contact method. Since this method for contact analysis is faster to solve, it should be helpful to analysts who work with large assemblies, where multiple parts are in contact with each other. The method becomes even more helpful when performing an optimization or a design of experiments (DOE) with a large number of runs.

Introduction

Finite element analysis has become one of the most popular numerical methods in the design process in various industries such as aerospace, automotive, rail, biomedical, consumer goods, defense, energy, electronics, heavy industry and marine over many years [1]. Moreover, with the improvements in finite element solvers and the access to better computational hardware it has become easier to use finite element analysis in the design processes of every part or assembly being manufactured. Simultaneously the design period for new parts has been continuously shrinking due to tremendous competition between different manufacturers. Due to this there is a constant need to reduce the finite element analysis time, in the design process of a part [2].

An important part in the finite element analysis process is the solver time. There are many factors which affect the solver time such as the size of the FE model, number of load-steps, non-linearity in the problem, etc. There can be three forms of non-linearity in a FE analysis problem: geometric non-linearity, contact non-linearity and material non-linearity. Geometric non-linearity occurs in a finite element problem when the deformation in the structure due to the applied loads is large, usually when it is more than ten percent of the element thickness [3]. Material non-linearity can occur in a finite element problem when the deformation in the structure due to the applied load is affected by the change in strain, temperature or pressure or other material property parameters. When two non-rigid bodies encounter each other, the geometry at the contact region changes till the force or stress in the system come to an equilibrium condition. This process introduces a nonlinearity which is dealt by contact elements. This type of non-linearity is called contact non-linearity. Presence of any of these non-linearity increases the solution time considerably.

Secondly, when large assemblies are being studied, it might become difficult to approximate the equivalent force acting on a part under investigation. This makes it necessary to consider all the components of an assembly or a sub-assembly during the design study of a part. The interaction between the parts in the assembly can be defined through rigid or non-rigid connections. Connections defined through contact interfaces are widely used in finite element analysis. The results and solution time of a finite element problem are highly dependent on the contact definition which is used. In cases, where we have no material non-linearity or geometric non-linearity, non-linear non-friction contacts can be used in the contact definition.

This paper is focused on reducing solution time due to contact non-linearity in finite element problems which have no material or geometric non-linearity. A finite element problem using non-friction contacts are compared, along with analytical solution for accuracy and time reduction. A new non-friction contact definition is introduced in this paper, which uses Altair's proprietary interpolation method for solving contacts: Fast Contact.

Hertzian Contact

Heinrich Hertz is widely associated with contact mechanics. Contact mechanics is the study of the deformation of two bodies which touch each other. Hertz studied the contact stress between two spheres touching each other in 1882. These stresses are also referred to as Hertzian contact stresses. In this paper, instead of taking two spheres, contact conditions will be simulated between a sphere and a plate. Stresses which occur at the contact interface are calculated theoretically and then compared with a finite element simulation which uses the Fast Contact formulation.

Theoretical Calculation

A sphere of radius R is assumed to be in contact with a flat plate indenting it. The sphere is made up of material stiffer than the plate. So, when a force F is applied on the sphere to push it against the flat plate, stresses are generated at the contact interface between the sphere and the plate. Following assumptions are made in this analysis,

1. Stresses are linear function of strain and within the elastic limit
2. Area of contact is much less than the size of the bodies
3. No friction occurs during contact [4-6]

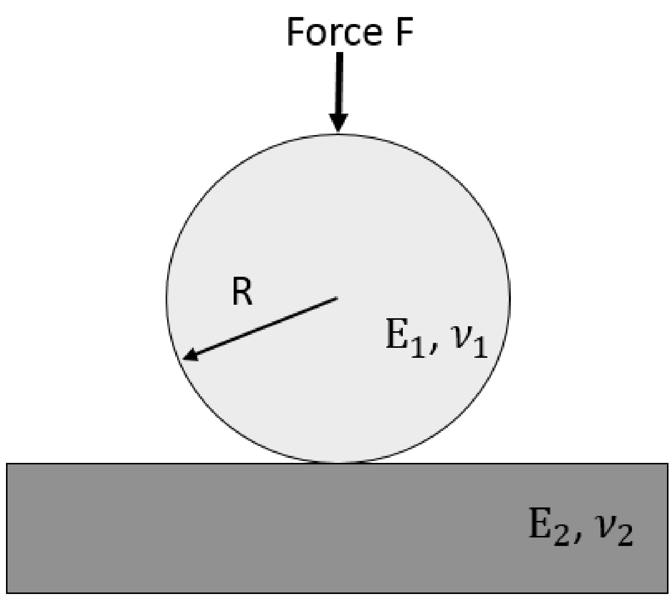

Figure 1. Figure showing a sphere loaded on a plate. Both have different material properties.

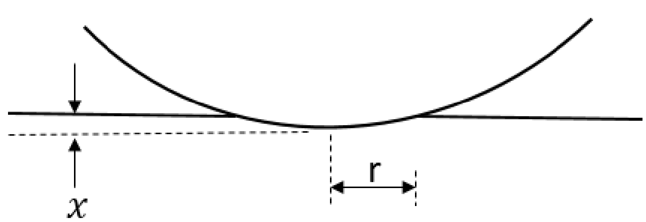

Figure 2. Zoomed image of the contact region between the sphere and the plate

The applied normal force $\boldsymbol{F}$ on the sphere creates a displacement $\boldsymbol{x}$ which can be expressed as

$$F = \frac{4}{3}E'R^{1/2}x^{3/2} \quad (1)$$

where

$$\frac{1}{E'} = \frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \quad (2)$$

and E_1 and E_2 are the Young's modulus and ν_1 and ν_2 are the Poisson's ratio for the sphere and the plate respectively.

Theoretically, the contact between a sphere and flat plate should be a small circular area or radius r. This is expressed as

$$r = \sqrt{R * x} \quad (3)$$

The contact radius can be expressed as function of the normal force by using Eq. (1)

$$r = \left[\frac{3FR}{4E'}\right]^{1/3} \quad (4)$$

Maximum contact pressure P is given by

$$P = \frac{2}{\pi}E'\left[\frac{x}{R}\right]^{1/2} \quad (5)$$

Expressing Eq. (5) in terms of normal force,

$$P = \left[\frac{6FE'^2}{\pi^3 R^2}\right]^{1/3} \quad (6)$$

Using eq. (6) maximum contact pressure will be calculated for a sphere of radius 20mm in contact with a plate of size 100mm x 100mm x 20mm. The sphere is made up of steel having a Young's modulus of 2.1×10^5MPa and a Poisson's ratio of 0.3. The plate is made up of aluminum and has a Young's modulus of 7×10^4MPa and Poisson's ratio of 0.33. A pressure of 1MPa is applied on the sphere. The plate is assumed to be constrained at the bottom. If the above information is substituted in Eq. 1.6, the contact pressure is calculated as 1278.128 MPa.

FEA Calculation

A quarter section of this model is created and solved in OptiStruct using Fast Contact and the results were found to be very close to the theoretical results. The model setup was done in HyperMesh and is shown below.

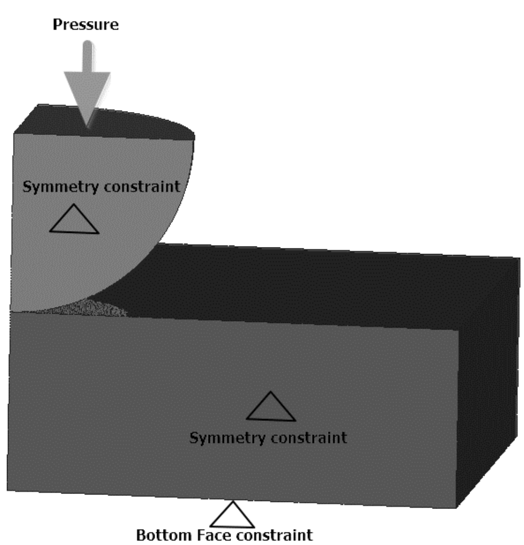

Figure 3. Symmetric quarter finite element model of sphere loaded on a plate

The results for contact pressure from OptiStruct are shown in figure 4.

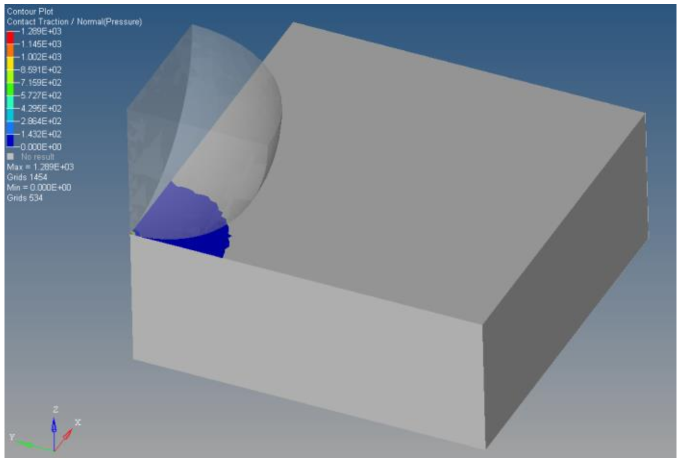

Figure 4. Figure shows contact stress results for a sphere loaded on a plate

Normal contact pressure using Fast Contact parameter was found to be 1289MPa. This is within one percent of the theoretical results. Definitely, if the mesh density is increased in the contact area, the results will be closer to the theoretical value of 1278MPa. Thus, it is seen that Fast Contact solution of OptiStruct gives correct results for a contact analysis.

Another factor to note is the time taken to solve the problem. Generally, problems involving non-linear contact take more time to solve than a linear analysis problem. In the above problem, it was seen that the solution of the same problem using Fast Contact in OptiStruct was around 12 times faster than the regular non-linear contact solutions. Table 1 shows that comparison between the different contact solutions. OptiStruct v14.230 was used with 12 cores to solve this problem. There is no friction involved in any of these contact simulations. Also, there is no material or geometric non-linearity in this model setup.

Table 1. Comparison of OptiStruct contact solutions for a sphere loaded on a plate

OptiStruct Contact solutions	Fast Contact	Non-linear Contact
Total number of Tet elements	127703	127703
Total number of Contact Elements	1403	1403
Maximum Normal Contact Stress (MPa)	1289	1289
Elapsed Time (min:secs)	1:18	15:31

During solving this problem, it was also observed that the solution time reduces proportionally with the contact area, if Fast Contact is used instead of non-linear contact solution. The main reason for this is the new solving algorithm which is used in Fast Contact.

Applications

This type of solution can be applied to any use case where an assembly is being analyzed for contact non-linearity without friction and where there is no geometric or material non-linearity involved. A wide variety of general finite element analysis problems from the rail industry and aerospace industry fall into this category.

One such use case of the aerospace industry is demonstrated ahead in this paper. A non-symmetric engine mounting bracket with a pin connection shown in figures 5 and 6, was analyzed for von mises stresses and contact stresses. Such engine mounting brackets support the aircraft engines and are connected to the aircraft wings.

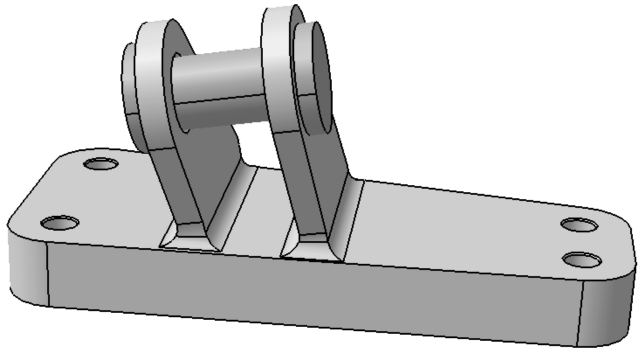

Figure 5. Non-symmetric aircraft engine mounting bracket with pin connection

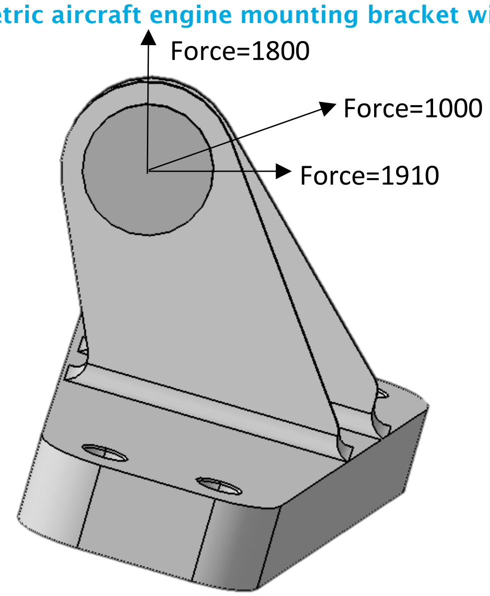

Figure 6: Side view of non-symmetric aircraft engine mounting bracket and the connecting pin showing the different loads acting on the pin

The bracket is made of aluminum and the pin is made of steel. Both have solid tetra mesh. The bolt holes of the bracket are constrained and the forces are applied on the pin along its horizontal and vertical axis. The results of the analysis are shown below.

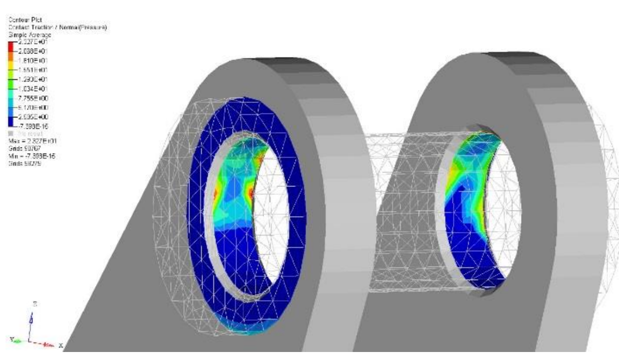

Figure 7. Figure showing the contact stresses between the non-symmetric aircraft engine mounting bracket and the connecting pin

Table 2. Comparison of FAST contact vs Full non-linear contact solutions for a non-linear static analysis of a pin in contact with a non-symmetric aircraft engine bracket

OptiStruct Contact solutions	Fast Contact	Non-linear Contact
Total number of Tet elements	281367	281367
Total number of Contact Elements	1152	1152
Maximum Normal Contact Stress (MPa)	23.27	23.27
Elapsed Time (min:secs)	1:06	12:16

From table 2, it is seen that the results for contact stresses are the same, but the time taken to solve this problem was drastically less with FAST contact. Also, in this case the peak von mises stress occurs at the fillets of the bracket for vertical and horizontal loads applied along the pin axis. This is the best-case scenario for considering FAST contact as it gives the quick and accurate result compared to a full non-linear contact solution.

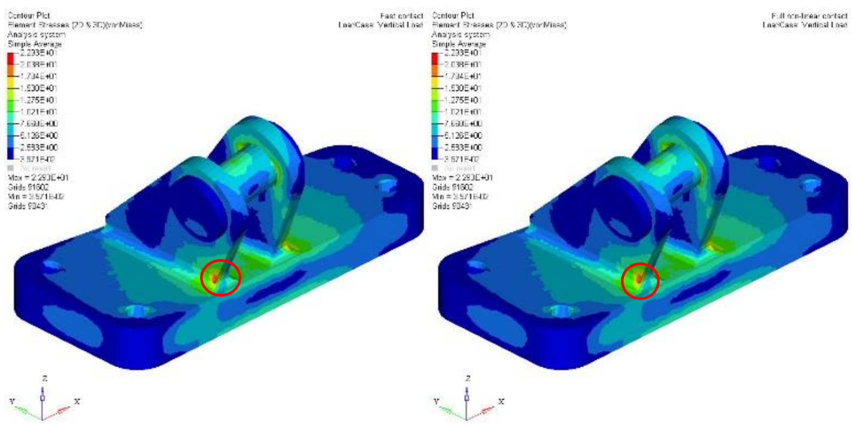

Figure 8. Color contoured figure showing the location of maximum von mises stress on the non-symmetric aircraft engine mounting bracket when a vertical load is acting on the connecting pin. Results on the left side are for FAST contact solution. Results on the right side are for full non-linear solution.

Often engineers working in the finite element analysis domain try to avoid creating the pin for brackets like above to avoid contact convergence issue. A workaround is to create flexible rigid body elements (RBE3) connecting the two centers of the bracket pin hole. Loads are applied at the center of the RBE3. But for non-symmetric brackets like the one shown in this discussion RBE3s will not give correct results if a cross load (or moment) is acting on the pin. Cross loads will try to create a moment and might cause a difference in loading at one of the shoulders of the brackets. This cannot be captured correctly using RBE3s. A result comparison for cross loads is shown in figure 9.

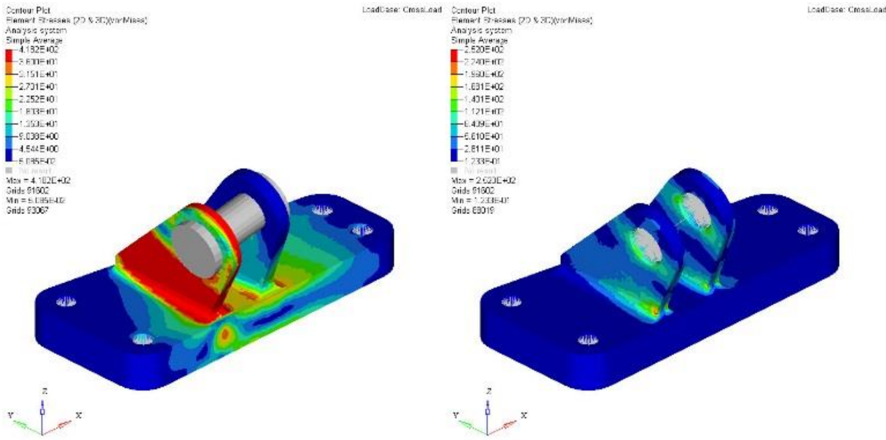

Figure 9. Color contoured figure showing the location of maximum von mises stress on the non-symmetric aircraft engine mounting bracket when a cross load is acting on the connecting pin. Results on the left side are for FAST contact solution with the actual meshed representation of the pin. Result on the right side show a RBE3-Beam representation of the pin connecting the engine bracket. Load is applied at the center of the beam.

When a cross load of 1000N is acting on the bracket in contact with a pin, maximum von mises stress of 41.82MPa occurs in one of the shoulders of the bracket. For the same load when an RBE3 element is created to simulate the same condition, the maximum stress is only 25.2MPa. This shows that for such load cases, the analyst should consider the pin in contact with bracket for correct judgement of the design.

Summary

Thus, it is seen that Fast Contact solution reduces the solving time of a non-linear contact problem by up to 12 times as compared to a full non-linear solution. However, users should make sure that there is no material or geometric non-linearity in such finite element problems. With optimization study and design of experiment study becoming more common in the design of parts, the use of Fast Contact becomes more advantageous. This will eventually reduce the design validation time thus resulting in faster design cycles.

References

1. “Practical Aspects of Finite Element Simulation”, Altair University, Sept. 2012
2. Praveen Yadav, Krishnan Suresh, “Large Scale Finite Element Analysis via Assembly-Free Deflated Conjugate Gradient”, Department of Mechanical Engineering, University of Wisconsin-Madison
3. Altair Hyperworks User Guide, http://www.altairhyperworks.com, 2016.
4. V.L. Popov, “Contact Mechanics and Friction”, Springer-Verlag Berlin Heidelberg 2010
5. Johnson, K. L, 1985, “Contact mechanics”, Cambridge University Press
6. K. L. Johnson and K. Kendall and A. D. Roberts, Surface energy and the contact of elastic solids, Proc. R. Soc. London A 324 (1971) 301-313

Contact Information

Numair Mazgaonkar, Altair Engineering Inc., 1820 E. Big Beaver Road, Troy, MI 48083. Email: nmazgaonkar@altair.com

Acknowledgments

Special thanks to Girish Mudigonda and Warren Dias from Altair Engineering Inc. for guiding me on technical aspects of this paper.