Fast Simulation Models for History Dependent and Non-Linear Time Consuming Finite Element Analyses

Problem presented by

Shashank Sinha

General Electric Company (GE)

Executive Summary

GE is looking for a fast simulation model for solid contact problems with friction that can be used to perform parametric studies in a practical way. Currently, GE relies on an ANSYS finite element model with Coulomb friction but the model requires several hours of CPU time per run. The study group determined that the simplest solution to this problem is to use parallel processing. A parallel finite element model was setup and run on one of the KAUST multiprocessor machines and produced significant speedups with very little effort allowing practical parametric studies to be performed. The study group also noted that additional optimizations both in the finite element model and in analytical or semi-analytical representations of the bodies in contact can produce additional speedups. This might be essential for three-dimensional problems.

Version 1.0
July 13, 2011
iii+5 pages

Report coordinator

G. Turkiyyah

Contributors

John Ockendon (University of Oxford, UK)
Tamer El-Sayed (KAUST)
Amir Siddiq (KAUST)
George Turkiyyah (American University of Beirut, Lebanon)
Arash Yavari (Georgia Institute of Technology, USA)

KSG 2011 was organised by
King Abdullah University of Science and Technology (KAUST)
In collaboration with
Oxford Centre for Collaborative Applied Mathematics (OCCAM)

Contents

1 Introduction

GE is looking for a fast simulation model for solid contact problems with friction that is independent of ANSYS (Finite Element Method software package used by GE). The simulator would take as input the loading history and return the contact tractions at the interface. It is desired to have a faster simulation than the current ANSYS code (which takes hours to run per simulation) in order to allow for meaningful and practical parametric studies to be performed [1].

2 Test Case

A specific test case of interest to GE is shown in Figure 1. The model is a two-dimensional discretization of a half-cylinder pushed into and dragged along a half-plane. Both the cylinder and the base are assumed to be linearly elastic with a Young's modulus $E = 125 \times 10^3$ and a Poisson's ratio $\nu = 0.3$. Coulomb friction is used to model the interface behavior, with a friction coefficient $\mu = 0.45$. The ANSYS finite element simulation uses about 30,000 linear quadrilateral elements to model the cylinder and the base. Contact elements are used to connect the perimeter nodes of the cylinder mesh to nodes on top of the base.

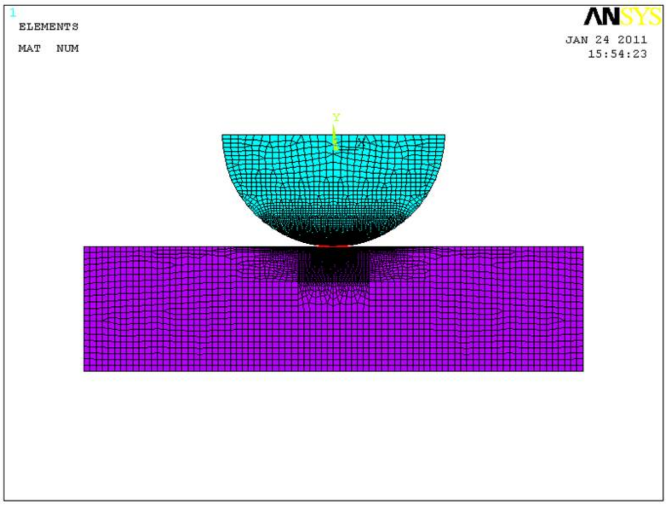

Figure 1: ANSYS model of the contact test problem

The cylinder is subject to the normal (P) and tangential (Q) loading histories shown in Figure 2.

3 Model Setup on a multiprocessor machine

It was determined that one of the easiest ways to get significant speedup for this simulation is to run it on one of the multicore machines available at KAUST. It took very little time to set up the simulation in another finite element package that runs

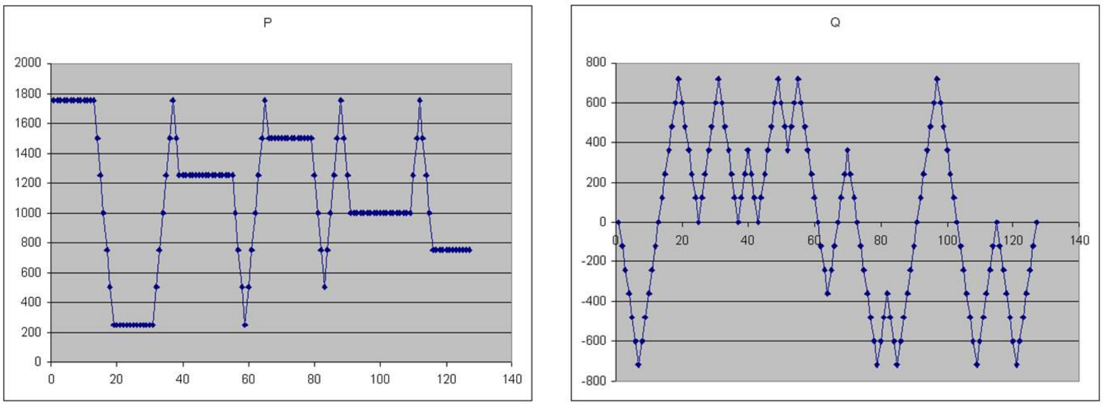

Figure 2: Normal and tangential force histories

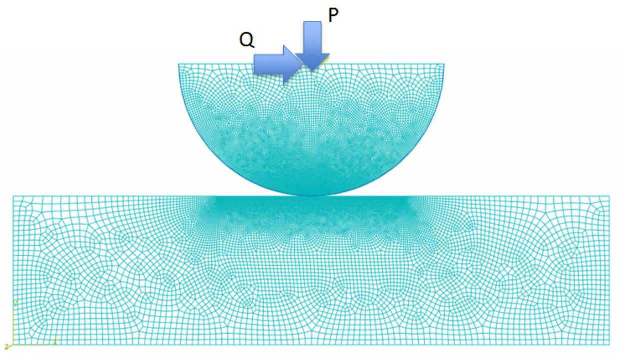

Figure 3: Discretization of the contact test problem

on multicore machines. The finite element model used linear quadrilateral plane strain elements and used the same level of refinement for reference. Contact was defined using a master-slave formulation with the same Coulomb friction coefficient as above. The model is subjected to the same load histories. See Figure 3.

The runtimes of the model using a varying number of cores is shown in Table 1. Using 8 cores, the runtime of the simulation was reduced by more than a factor of 2. Additional cores did not yield additional speedup. It appears that the solution methods of the finite element software are not scalable and that the communication overhead between the cores and the serial portions of the code are limiting additional speedup.

Number of cores	Time (min)
1	38
8	17
16	39
24	57

Table 1: Computation time of the simulation on multiple cores

In looking at the displacement response history, the study group noted that there is tangential slip along the interface but that the slip changes significantly along the contact region. Figure 4 shows the relative tangential slip at various points in the response history for the loading of Figure 2.

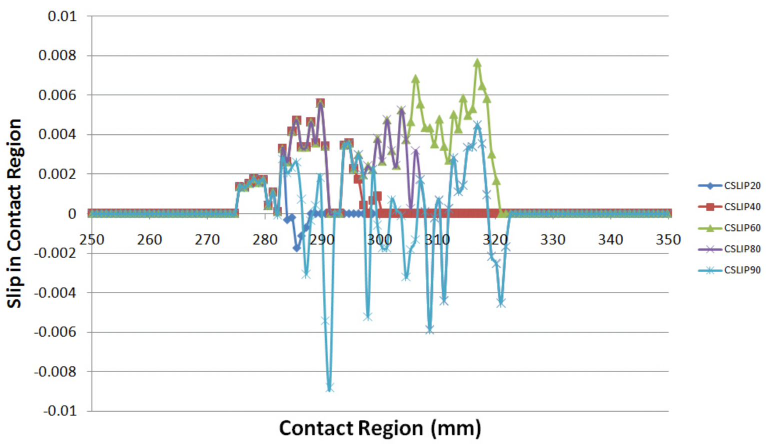

Figure 4: Tangential slip at various times

For the problem at hand, off-the-shelf commercial finite element packages can make the contact simulations of the sort described in the test case practical for engineering use. However the study group noted two additional optimizations to improve the runtime behavior of the model:

- Decrease the mesh size. The level of refinement in the contact region appears to be excessive and a smaller number of nodes can provide the necessary accuracy. Since the computational effort grows superlinearly with the number of nodes, this could result in substantial savings.
- The geometric computations involved in checking pairwise contact between elements can be optimized. However this will require writing a new solver as commercial finite element codes do not provide user-level access to these internal computations.

4 Analytical Model

For this two-dimensional problem, the study group did not feel that an analytical model is needed as it is straightforward to set up and run such simulations using standard off-the-shelf commercial finite element codes. In addition, the numerical finite element models have the advantage that the incorporation of more sophisticated constitutive models of material as well as interface behavior can be readily done. However, if 3D problems are to be simulated, then a straightforward discretization as was done here will not likely result in practical computations even on large multicore machines. In this case, the study group discussed that the half space may be modeled using an integral equation approach resulting in a formulation that only uses the surface nodes of the half space. Purely analytical solutions are also possible [2]. However these ideas were not pursued further but could be explored if there is interest in faster simulations or in solving three-dimensional problems.

5 Hysteretic Behavior

Also of interest was the system's hysteresis loop which can be used to estimate the contact stiffness. Such characterizations are often done experimentally, using devices such as the one shown in Figure 5, which results in hysteretic loops as shown in Figure 6, from which the contact coefficients may be estimated. For the loading of the current problem, the resulting hysteresis generated by the simulation is shown in Figure 7.

Figure 5: Experimental apparatus for determining contact coefficients

6 Conclusion

Solving the 2D contact problem using finite element discetization with standard master-slave contact proved to be quite effective and straightforward using the computational capabilities available at KAUST. More sophisticated material and interface models can also be readily incorporated if desired. Three dimensional problems will however require different formulations in order to make them practicable for routine use in engineering offices.

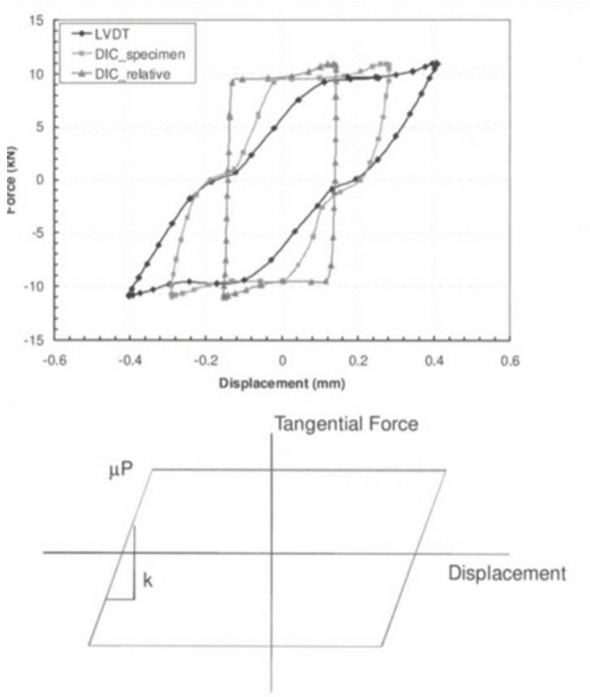

Figure 6: Hysteretic behavior

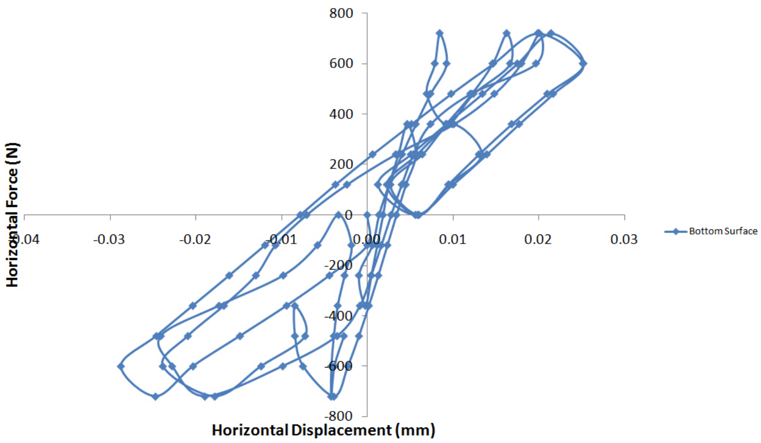

Figure 7: Simulated Hysteresis

Bibliography

[1] P. T. Rajeev, H. Murty, T.N. Farris, *Load-History effects on fretting contacts of isotropic materials*. Journal of Engineering for gas turbines and power. April 2004, Vol 125, pp385.

[2] Johnson, K. L, 1985, *Contact mechanics*, Cambridge University Press