Simulation of a Parking Pawl Mechanism with ABAQUS/Standard and ABAQUS/Explicit

Das Ramnath[a], Prashant Andrade[b]

[a] Ford Motor Company [b]Hibbitt, Karlsson & Sorensen (Michigan), Inc.

Abstract: This paper describes some analyses of a parking pawl mechanism using the ABAQUS finite element analysis code. In particular, we focus on obtaining the torque to turn curves for a given Park Reverse Neutral Drive Lever ("PRNDL") input. Because of the dynamic energy release events in the course of the simulation, a solution is not possible with a standard nonlinear static analysis. Various alternatives are examined. Among these are the addition of a stabilizing dashpot, automatic stabilization, use of contact-related damping with ABAQUS/Standard and an explicit dynamics solution using the rigid-to-rigid penalty contact capabilities of ABAQUS/Explicit. We compare the solutions obtained by the different approaches and make recommendations for how ABAQUS might be enhanced to handle this class of problems even more efficiently.

1. Introduction

Automatic transmissions have been a fairly common option on most automobiles since the late 1940's. In these transmissions, the driver is provided with a gearshift lever that can be moved between different gear positions. The gear positions typically available on standard modern automatic transmissions are Park, Reverse, Neutral, Drive and gears 1 through 3. When the gearshift is placed in Drive, for instance, a manual valve directs fluid to the clutch packs that activates first gear, and sets up the system to monitor vehicle speed and throttle position so that it can determine the optimal time and the force for the 1-2 shift. There are both ergonomic and safety concerns in designing for the PRNDL force input required to switch between gears. For example, the input required for the driver to shift from Park to Reverse should be low enough to feel comfortable, yet not too small since this can cause the system to inadvertently move from the park position to the reverse position resulting in dire consequences. The magnitude of the input required to shift between other gears should not exhibit too much variation. As demonstrated in this paper, the ABAQUS finite element code provides an accurate and efficient way to simulate the mechanism and obtain information to address the above design concerns.

An assembly consisting of a cam, a pin with a preloaded spring and a manual valve consisting of a piston sliding in a valve body is used to translate the PRNDL input into a gear change. An ABAQUS model of the components used in the analysis is shown in Figure 1.

The rooster comb cam contacts a pin as shown in Figure 1. The cam itself has two attachment points. The first attachment point at the top is to the gearshift handle. The second attachment point on the cam is to a manual valve. The cam translates the PRNDL input into the correct position of the manual valve in the valve body. The manual valve opens the relevant hydraulic circuit to actuate the transmission. The pin slides on the surface of the rooster comb; the detent spring to which it is attached provides the resistance to motion. The pin resting in the valleys between the peaks of the cam profile represent various gear selections.

The center of the rooster comb is keyed to a shaft that is supported in the transmission case. We denote the reaction torque required at the level of this shaft to rotate the cam through the various gear positions as the torque to turn curve, the key response used to make design decisions. As mentioned above, the torque to turn curve obtained as the driver moves the position of the lever should possess characteristics like a low peak from Park to Reverse while not being too small to allow for a move from the Park position to the Reverse position due to a small accidental input. The peak torque should also exhibit a rounded profile to ensure a better "feel". It is also important, that the pin remains in the valley when the driver does not actuate the PRNDL. Although, this simulation is not conducted here, the idea is tantamount to releasing the pin from a peak and ascertaining that the pin does not overshoot the next peak.

2. ABAQUS model relevant details

The goal of the analysis was to obtain torque to turn curves for a static rotation of the rooster comb and evaluate the stress state in the detent spring for design purposes.

In this section we describe the ABAQUS model. A zoomed in view of the pawl mechanism is shown in Figure 2.

The initial geometry has the pin surface defined such that the pin geometry overlaps the rooster comb mesh. This corresponds to the undeformed position of the detent spring. The analysis consists of preloading the detent spring and then rotating the rooster comb about the axis of the transmission case attached shaft to move from reverse to the park position. Next, the sense of the rotation of the rooster comb is reversed so that the pin moves from park to reverse through neutral, drive and the other gears.

ABAQUS/Standard offers a convenient and reliable way to specify contact and kinematic constraints between components in an assembly. In the current simulation:

- The pin analytical rigid surface contacts the detent spring with a “tied contact” condition.
- The rooster comb surface contacts the pin surface.
- A rigid body, with a reference node on the axis of the transmission case shaft, enforces the revolute constraint of the rooster comb about the shaft axis.

The ABAQUS analysis is divided into six steps. The time period and description for each of the steps is given below.

The first four steps of the analysis preload the detent spring with a suitable application of boundary conditions and by disabling and enabling contact with the `*MODEL CHANGE` option. The rooster comb is then rotated in Steps 5 and 6 to obtain the torque to turn curve.

- **Step 1**, Time 0.0 to 1.0: Deactivate contact between the pin and the rooster comb teeth.
- **Step 2,** Time 1.0 to 2.0: Deflect the detent spring with a displacement boundary condition on the pin reference node
- **Step 3**, Time 2.0 to 3.0: Activate contact between the pin and the rooster comb teeth
- **Step 4**, Time 3.0 to 4.0: Release the boundary conditions on the pin reference node and establish contact between the pin and the rooster comb teeth
- **Step 5**, Time 4.0 to 5.0: Rotate the rooster comb, via the reference node on the axis of the transmission case, to shift from park to reverse (negative rotation about global degree of freedom 6 in Figure 1).
- **Step 6**, Time 5.0 to 6.0: Rotate the rooster comb to shift from reverse to park (positive rotation about global degree of freedom 6 in Figure 1) and then through all the other gears.

3. Solutions to obtain the torque turn curve

3.1 Introduction

The problem studied is straightforward except for the inherent points of dynamic instability, the so-called “blow-by” points. It offers an opportunity to examine various techniques ABAQUS provides for obtaining accurate solutions despite such instabilities.

A preliminary model of the system had the pin defined as a discretized rather than an analytic rigid body. As expected, switching to an analytic body enhanced the convergence and reduced faceted contact-related noise substantially. The analysis was first attempted with the default Newton-Raphson incremental iterative technique—a typical nonlinear *STATIC analysis in ABAQUS/Standard. Not surprisingly, convergence problems were encountered with this approach during Step 5. This stemmed from the presence of a dynamic instability. The detent spring attached to the pin stores up strain energy as the pin traverses the teeth of the rooster comb. As the pin wants to traverse the tip of each tooth, there is a sudden release of some of the strain energy stored in the spring. This blow-by effect is a dynamic phenomenon: the solution is outside the radius of convergence of the static Newton solution, since the tangent stiffness matrix does not include the stabilizing mass terms.

3.2 ABAQUS solutions

3.2.1 Implicit dynamic solution

A successful simulation was obtained by using the implicit dynamics capability of ABAQUS. Since there is negligible damping in the system (due to the ALPHA factor in the *DYNAMIC time integration scheme), the dynamic solution at the blow-by points was noisy. Also, a softened contact relationship had to be used for the normal interaction between the pin and rooster comb surface to obtain a solution. In this case, ABAQUS/Standard does not use the impact algorithm, which destroys kinetic energy of the nodes on the surface when impact occurs, but instead assumes a perfectly elastic collision. The ABAQUS/Standard User’s manual (Section 21.3.3, V6.2) states that softened contact may work well in dynamic calculations where impact effects are not important, for example, if contact changes are primarily due to sliding motion along a curved surface. This fits the problem at hand when the pin is not close to rolling off the tip of a tooth. The consequence of using softened contact is that the slave nodes bounce back immediately after impact with the master surface; hence, extensive "chattering" results, leading to significant time-step cutbacks and, therefore, many time increments. Despite the solution being obtainable here, pursuing implicit solutions to non-linear dynamics problems involving impact is in general, not recommended.

In addition, inertial effects are unnecessary in this case, since the goals of the analysis do not require calculating impact forces or other dynamic effects. The designer is looking for a solution to the quasi-static problem. With this in mind, some other more effective approaches were examined.

3.2.2 Solution by addition of a dashpot element

A very useful feature of ABAQUS is the ability to introduce dashpot elements to counter instability even in a static analysis. This works well in concert with the automatic time incrementation scheme used by ABAQUS. In the problem studied, the source of the instability was evident. To damp the motion of the pin as it slipped off the tip of a tooth of the rooster comb, we connected a single dashpot element in the 2 degree of freedom between ground and the reference node of the pin rigid body to help dissipate some of the energy released by the detent spring and stabilize the tangent stiffness matrix. The coefficient used for the dashpot used was 0.0001 (N-sec/mm). Ideally, one should make the dashpot coefficient as small as possible. In this case, there was little sensitivity to the solution for dashpot coefficients between 0.1 and 0.0001. The torque to turn curve obtained is given in Figure 3. All torque to turn curves presented below have been normalized for confidentiality purposes. There are repeating pairs of zero torque intercepts in Figure 3. These correspond to times when a) the pin reaches either a peak or a valley of the rooster comb teeth, The direction of application of the reaction torque also changes in each of these cases due to the resultant of the contact pressures on the rooster comb acting in opposite directions. Also it is noticed that the torque to turn plot is smooth when the pin transfers the load from a tooth to a neighboring tooth (in a valley). This is because the pin is designed to always remain in contact with one of the rooster comb teeth it is wedged between.

It is strongly recommended that when stabilizing a solution using dashpots, a comparison be made between the viscous dissipation energy (ALLVD) and the internal energy (ALLIE) of the relevant components to judge the influence the stabilizing dashpots have on the solution. ALLVD and ALLIE plots for the value of the dashpot used are given in Figure 4. Clearly, the viscous dissipation energy is extremely small compared to the internal energy stored in the detent spring and the rooster comb, indicating that the dashpots have done little to change the fundamental solution, but have improved the radius of convergence of the tangent stiffness matrix.

3.2.3 Solution using automatic stabilization

ABAQUS offers the option to stabilize problems by applying damping throughout the model in such a way that the viscous forces introduced are sufficiently large to prevent instantaneous buckling or collapse but small enough not to affect the behavior significantly while the problem is stable. Often ABAQUS analysts use the `STABILIZE` parameter on the `*STEP` card as a first resort to overcome nonconvergence. Using this technique, with the default stabilization factor, leads to the torque to turn curve shown in Figure 5. It is seen that the torque required is significantly bigger than the dashpot solution obtained earlier. Figure 5 indicates the danger of using the stabilize parameter without due attention to performing checks to evaluate the quality of the solution. A comparison of the stabilization energy (ALLSD) and the internal energy (ALLIE) is shown in Figure 6, it can be seen that the stabilization energy dominates the internal energy for most of the time duration. Thus the gross error of the torque to turn curve in Figure 5 can be attributed entirely to an inappropriate use of automatic stabilization due to an unsuitable default factor for this application.

The default factor used for stabilization is printed in the data file. The results of reducing the stabilization factor to one hundredth of the default are shown in Figure 7. It can be seen that the solution obtained using this lower factor gives a torque to turn curve closer to the solution using a dashpot element. The stabilization energy (ALLSD), however, is still quite substantial compared to the internal energy (ALLIE) during Steps 5 and 6 as shown in Figure 8. Monitoring the energies carefully to evaluate solution quality while using the stabilize parameter and checking the viscous forces (ABAQUS output variable VF) and comparing them with the expected nodal forces to make sure that the viscous forces do not dominate the solution is imperative when using the stabilize parameter. Because of the sensitivity of the solution o the stabilization factor, this approach was determined to be less appealing than the single dashpot approach described above for this application.

3.2.5 Solution using contact damping

ABAQUS also has the ability to add viscous damping to the behavior of a contact pair rather than to all the nodes in the model via the *CONTACT DAMPING option. The viscous forces applied are normal to the master surface and are proportional to the relative approach velocity of the two surfaces in the pair. Two analyses with damping coefficients of 0.002 N-sec/mm and 0.0002 N-sec/mm were run; there was no appreciable difference in the torque to turn curves obtained by ABAQUS, thus giving us confidence that the damping was not influencing the model in the stable regime. The torque to turn curve ratio obtained is shown in Figure 9, with the solution obtained using the dashpot element superimposed. It is seen that the torques required are very close to the dashpot solution obtained earlier. A comparison of the viscously dissipated energy (ALLVD) and the internal energy (ALLIE) is shown in Figure 10, it can be seen that the total viscously dissipated energy at the end of the analysis is about a fifth of the average internal energy over the course of the analysis. This approach is as appealing as the single dashpot method.

3.2.4 Solution using ABAQUS/Explicit

Rigid-to-rigid contact capabilities of ABAQUS/Explicit can also be used for this application. In this case the rooster comb can be declared to be a rigid body. The rigid-to-rigid contact capabilities of ABAQUS/Explicit were used to model contact between the rooster comb surface and the pin. A penalty contact stiffness value governing normal contact interactions between the rigid surfaces can be set. A penalty contact stiffness value of 10000 N/mm^2 was chosen based on the second author's experience with similar ABAQUS/Explicit models and specified after setting the `PRESSURE-OVERCLOSURE` parameter to `LINEAR` on the `*SURFACE BEHAVIOR` keyword line.

All of the implicit solutions above were run within a few hours on a typical workstation. We wanted to ensure that any potential explicit solution remained competitive with the implicit solutions. To this end, for the stable time increment corresponding to the mesh discretization of the detent spring shown in Figure 1, we tried to choose time scaling values (100 milliseconds for each of Steps 5 and 6 of the analysis) reasonable enough for the analysis to complete within a day on a typical workstation. However, with these constraints, the time scaled solution was very noisy due to the high frequency vibrations of the detent spring and the changing dynamic contact conditions.

To eliminate the deleterious effects of these vibrations we replaced the spring by an effective stiffness matrix, as described next. The spring and the tied pin were isolated in a separate model. The spring was preloaded to three different levels with nonlinear geometric effects (displacements of the pin of 0, 2.5 and 5 mm in the global negative 2 direction) representing the expected range of motion of the spring. Perturbation steps following each general loading step were run to calculate the stiffness matrices of the spring at the pin location about each preloaded state of the spring. Since the stiffness matrices obtained about the three different preloading conditions were relatively close, an ad hoc scheme of using the average of the three stiffness matrices was tried to see if useful results could be obtained. This procedure can be loosely thought of as applying a filter to ignore the frequency content of the spring and represent only its static stiffness. The step sequences specified was the same as the ABAQUS/Standard run. The stiffness of the detent spring was specified as a combination `CARTESIAN` and `ROTATION` connector element, with the spring stiffnesses being specified by using the `COUPLED` parameter on the `*CONNECTOR ELASTICITY` keyword line. A penalty contact stiffness value of 10000 N/mm^2 was chosen again. ABAQUS/Explicit does not have a basis to calculate the stable time increment for this model since the rooster comb has been declared rigid and the detent spring is represented by spring stiffnesses. We choose a fixed time increment of 5 milliseconds and set the step time for each of Steps 5 and 6 to be 500 milliseconds. This resulted in stable analyses with fast runtimes of less than an hour on a workstation. ABAQUS/Explicit analyses with a large number of increments (> 300000 as per the ABAQUS/Explicit User's manual) should always be run in double precision. Although the above analysis took on the order of a hundred thousand increments, we noticed that the history output traces of quantities like reaction forces and moments were substantially noisier for a single precision run and we use the results of the double precision run in reporting the results.

(Diehl, 1999) discusses the importance of using Digital Signal Processing (DSP) to improve the interpretation of results obtained using ABAQUS/Explicit. Although our system did not contain elastic deformable bodies undergoing impact (the subject of Diehl's paper was the analysis of drop tests on pagers and cell phones), the penalty contact stiffnesses effectively introduces elasticity as an ingredient into the solution. With no dissipation mechanism, high frequency noise is expected to affect the system and the data has to be filtered to obtain intelligible results. This high frequency noise is indeed observed in our reaction force and moment plots. Following the advice in Diehl's paper, we ensured that the reaction torque of interest at the rooster comb rotation boundary condition application point was stored at every time interval of the explicit analysis to prevent aliasing effects. We then used the sbFilter (Sine-Butterworth filter) tool provided with the ABAQUS/Viewer XY data processing toolkit to filter the data with a cut off frequency of 500 Hz. The filtered results are plotted along with the dashpot solution of ABAQUS/Standard in Figure 9. This solution can be seen to match the ABAQUS/Standard solution with dashpots well, given the nature of the assumptions made in obtaining the solution. It also indicates that the elasticity of the rooster comb is not a significant parameter in the problem and for practical purposes the rooster comb can be treated as a rigid body. This was also separately verified by boosting the modulus of elasticity of the rooster comb material by a factor of a hundred in the dashpot solution, the same torque to turn curve was recovered for the analysis with the stiffer rooster comb.

4. Alternative solution proposals

A potential solution to the problem that cannot be performed in ABAQUS at the time of writing of this paper would be an implicit static solution that allowed contact between rigid bodies with a penalty contact algorithm. The ABAQUS/Explicit solution and the ensuing check with a higher modulus for ABAQUS/Standard demonstrated that the elasticity of the rooster comb has very little bearing on the results of interest. The implementation of penalty contact in ABAQUS/Standard allowing contact between rigid bodies will enable us to perform this analysis without having to approximate the detent spring as a point system as we needed to do in ABAQUS/Explicit. There are obviously no wave propagation effects in a static analysis unlike those observed in ABAQUS/Explicit and this static rigid-to-rigid contact approach with ABAQUS/Standard could potentially lead to the most efficient solution approach. Indeed, the first author models the rooster comb as a rigid body in the simulations he has run for this problem with the ADAMS multibody simulation code. However, the solution obtained by ADAMS is a transient dynamics solution. The stabilizing dashpot will of course still have to be used in such an ABAQUS/Standard rigid to rigid contact run as the dynamic energy release events will still be encountered.

5. Conclusions and Recommendations

In this paper we have examined various successful approaches for simulating a parking pawl rooster comb mechanism with ABAQUS. Additional considerations to improve the radius of convergence of a standard incremental-iterative Newton-Raphson approach were required due to dynamic energy release events in the course of the analysis. The approaches studied were the addition of a stabilizing dashpot, automatic stabilization and contact damping with ABAQUS/Standard and an explicit dynamics solution using the rigid-to-rigid penalty contact capabilities of ABAQUS/Explicit. The solutions obtained by the different approaches were compared. The simple ABAQUS/Standard dashpot element and the contact damping solutions were found to be the most appealing with the current capabilities of the code. The ABAQUS/Explicit solution with the detent spring represented by an approximate stiffness matrix and the rooster comb declared to be a rigid body had the fastest run time but involved significant approximations and substantial ground work in setting up.

Based on our arguments in Section 4, we make the recommendation that HKS consider implementing penalty contact algorithms in ABAQUS/Standard for enabling contact to be considered between two rigid bodies. The authors expect to see a demand for this grow especially due to the increased use of the analysis codes for kinematic and kinetic analysis with the recently introduced general connector element functionality for the imposition of kinematic constraints and joint actuation.

In addition, we make the recommendation that HKS consider allowing the use of smooth CAD geometry representations of rigid surfaces directly for contact computations in both ABAQUS/Standard and ABAQUS/Explicit. Fortunately for the problem studied, the surface of the pin could be modeled with a simple analytical rigid surface. However, in general surfaces do not have such a simple representation. We feel that capturing this smoothness will make for more robust contact models involving rigid surfaces in ABAQUS/Standard analyses as well as lead to smoother results for ABAQUS/Standard and ABAQUS/Explicit contact analyses involving rigid surfaces.

Interpreting the torque to turn curves obtained (see Section 3.2.2) was relatively straightforward for this problem. However, this process would have been more convenient had ABAQUS/Viewer been able to display an animation of the motion in a window with a simultaneous display of a plot window having a visual marker point moving along the torque to turn curve in tandem with the motion animation. We recommend that HKS consider implementing this feature to match the functionality offered by most other multibody dynamics simulation codes.

6. Acknowledgements

The first author would like to thank Ford Motor Company for its encouragement and support.

The second author would like to thank Mark Bohm, General Manager, Hibbitt, Karlsson & Sorensen, Michigan, for his editorial and technical suggestions. Thanks are also due to Brad Heers at Hibbitt, Karlsson & Sorensen, Michigan for an explanation of the mechanism centered around a short virtual visit to howstuffworks.com. Many thanks are due to the other managerial, technical and administrative staff at Hibbitt, Karlsson & Sorensen (Michigan) for their support and encouragement.

7. References

1. Diehl, T., Carroll, D., Nagaraj, B., Motorola, "Using Digital Signal Processing (DSP) to Significantly Improve the Interpretation of ABAQUS/Explicit Results," ABAQUS Users Conference, Chester, United Kingdom, May 1999, pp. 191 - 212.

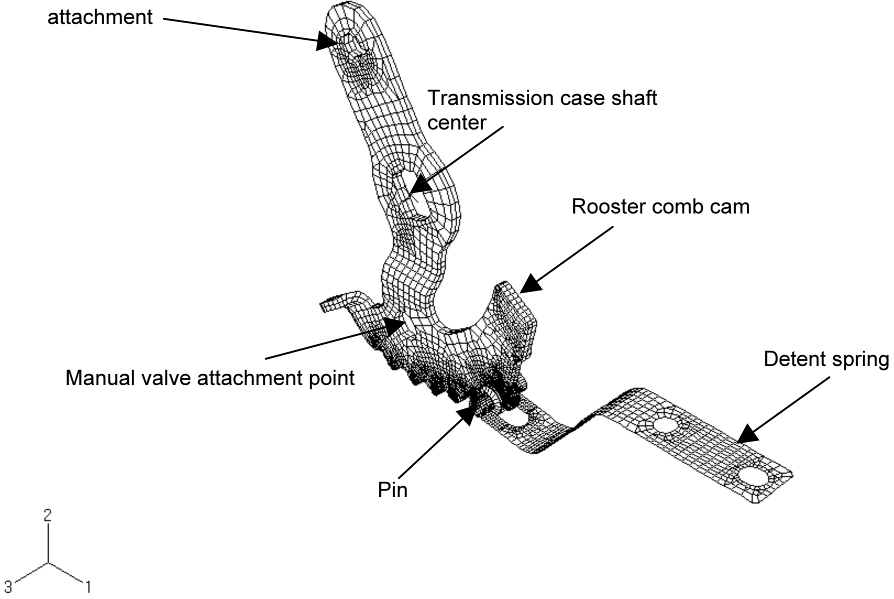

Figure 1. Parking pawl mechanism

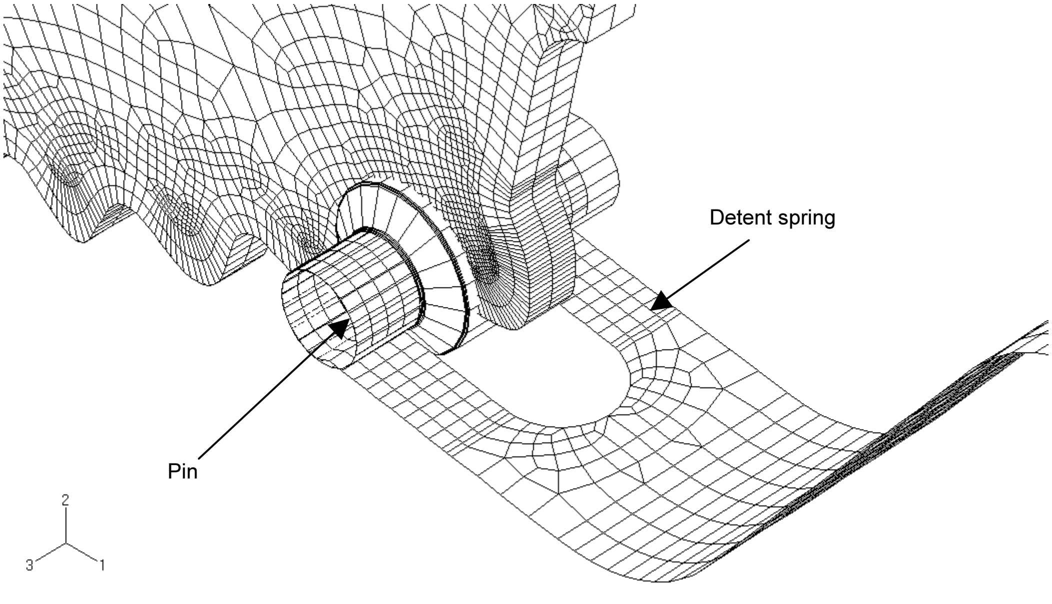

Figure 2. Zoom view of pin and detent spring

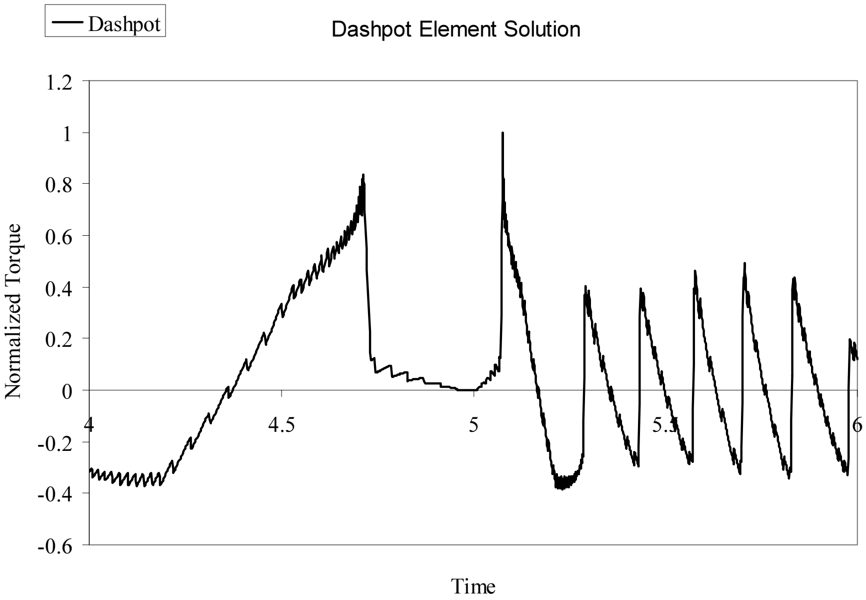

Figure 3. ABAQUS/Standard dashpot element solution

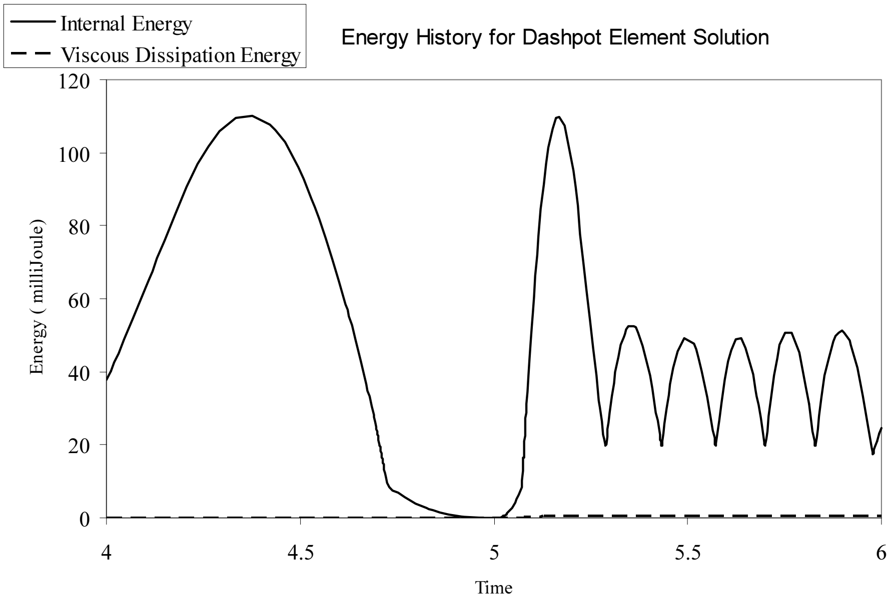

Figure 4. Energy history plot for dashpot element solution

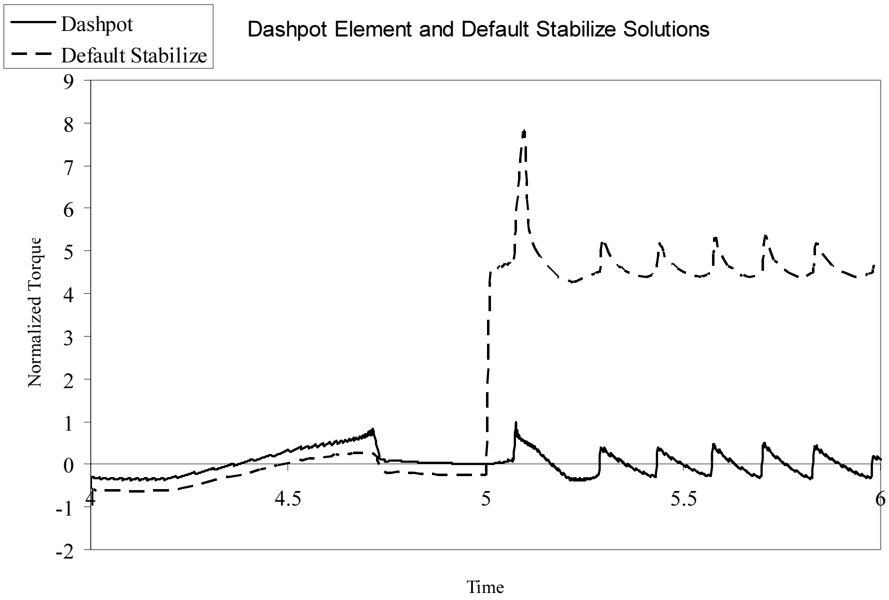

Figure 5. ABAQUS/Standard default factor stabilize solution

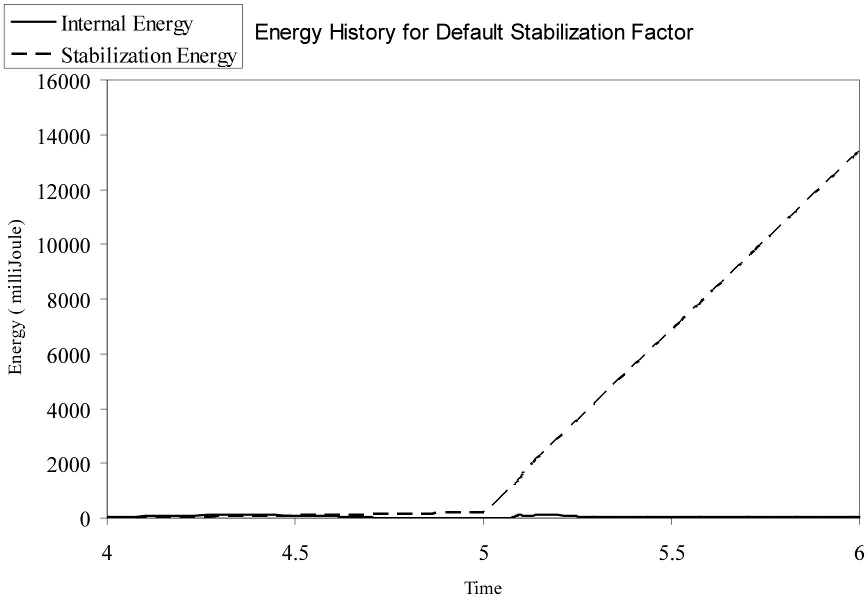

Figure 6. Energy history plot for default factor stabilize solution

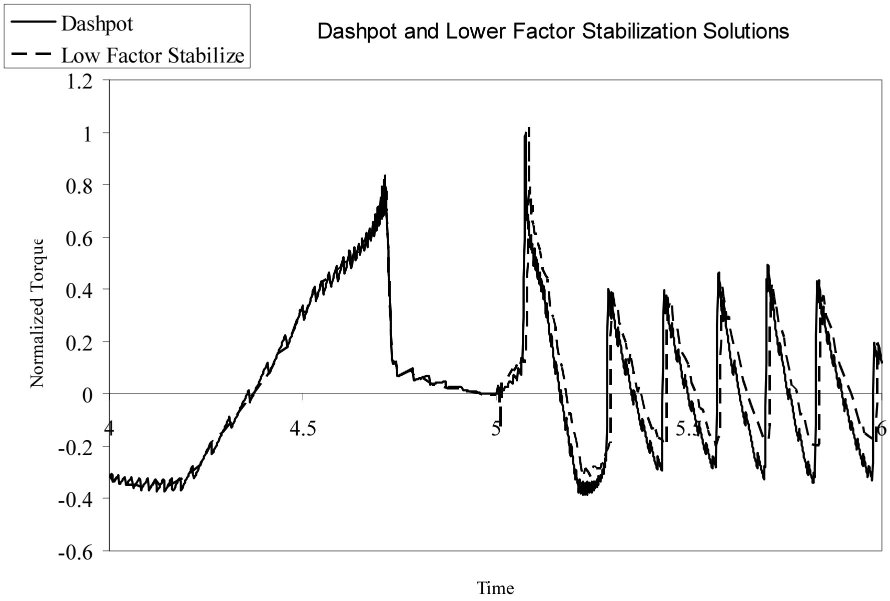

Figure 7. ABAQUS/Standard lower factor stabilize solution

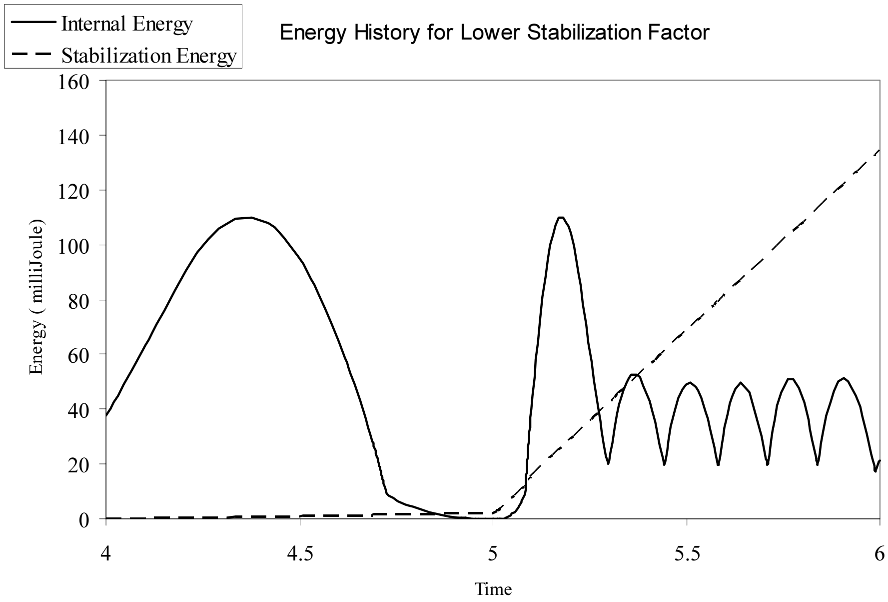

Figure 8. Energy history plot for lower factor stabilize solution

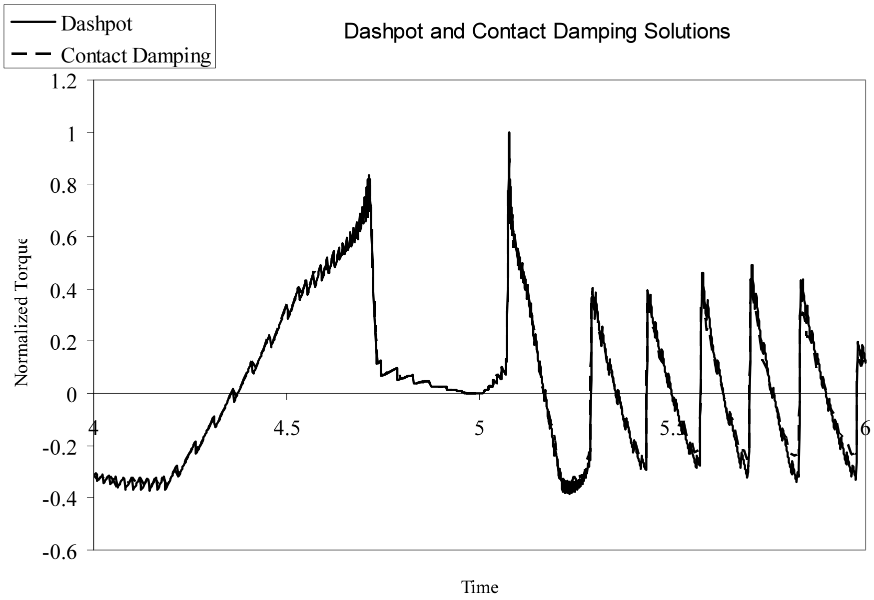

Figure 9. ABAQUS/Standard contact damping solution

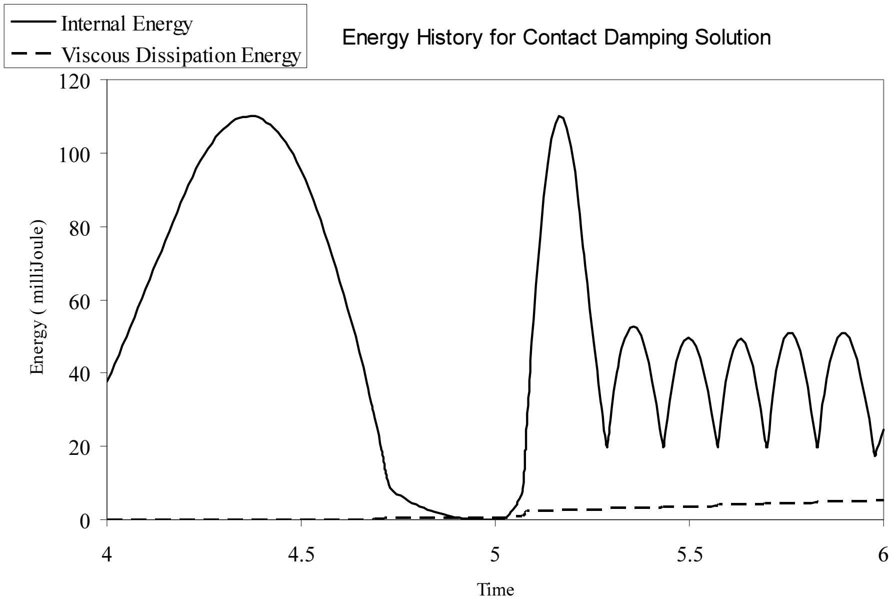

Figure 10. Energy history plot for contact damping solution

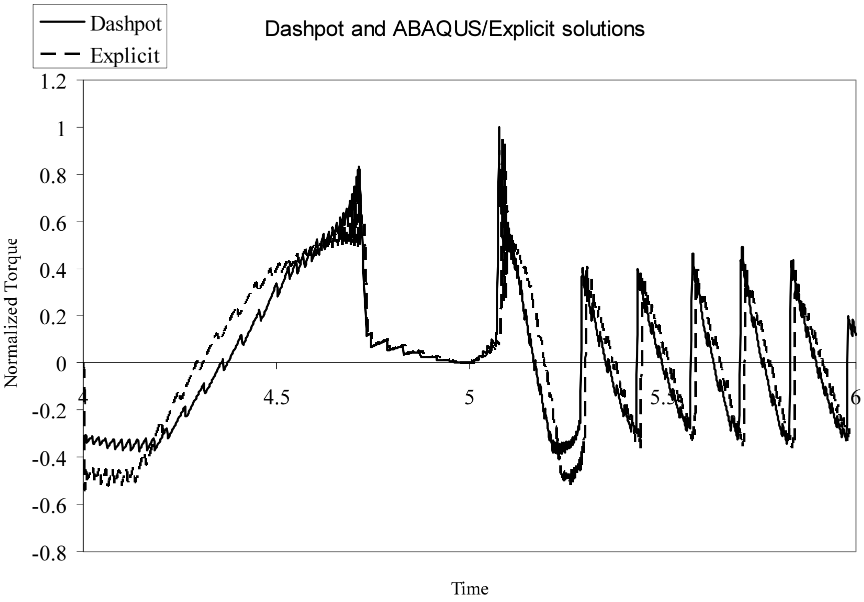

Figure 11. ABAQUS/Explicit solution