

Realistic Simulation of Golf Ball Impact

Xiaohu Liu, David Quinn, and Jorgen Bergström

Veryst Engineering, LLC, 47A Kearney Rd, Needham, MA 02494

Abstract: *This paper presents an Abaqus/Explicit simulation of the impact event when a golf club hits a golf ball. This is a challenging subject for finite element simulations because of its high strain rate behavior. The simulation emphasizes accurate material characterization and realistic model construction. Specifically, an advanced nonlinear viscoelastic material model from Veryst Engineering's PolyUMod library with strain rate dependence is calibrated with high-rate testing data to accurately capture the highly nonlinear behavior of the golf ball materials during impact. At the same time, a detailed finite element model of the golf ball is constructed with multiple layers of structure. The complex dimple pattern on the ball cover as well as the grooves on the golf club are modeled, both potentially important factors in impact response. Simulation results with the advanced material model are compared to those with a simple linear viscoelastic model to demonstrate the importance of accurate material characterization.*

Keywords: *Impact Simulation, Golf, Constitutive Model, High Strain Rate, User material, UMAT, VUMAT.*

1. Introduction

The competitive nature of the sports industry pushes the limit for better product designs. In the case of the golf industry, both ball and club manufacturers strive for new products that will give their players the edge in playing the game. The impact event when a golf club hits a golf ball is a big part of the game and plays a key role in the design of both golf clubs and golf balls. However, it has been a challenging subject for finite element simulations because of its high strain rate characteristics: the impact occurs within milliseconds and the golf ball experiences very large deformation during this period because of the polymer material from which the ball is constructed. The constitutive model used in the finite element model has to be able to capture the behavior of the material within a very wide range of strain rates.

In this paper, Abaqus/Explicit is utilized with the aim to produce a realistic simulation of this impact event. Emphasis is put on accurate material characterization and realistic model construction. Specifically, from Veryst Engineering's PolyUMod™ library, advanced nonlinear viscoelastic material model with strain rate dependence is calibrated with high-rate testing data to accurately capture the highly nonlinear behavior of the golf ball material during impact. The

following sections discuss the various aspects of the finite element model, including geometry, material model, impact simulation and results.

2. Geometry

The golf ball modeled in this study has a geometry that is based on the USGA/R&A Calibration model golf ball by Bridgestone Sports. It is of a two-piece construction that has a nominal diameter of 42.72mm, a core diameter of 38.5 mm and a cover thickness of 2.1mm [1]. The dimple pattern is based on a geodesic icosahedral pattern [2]. A snapshot of the golf ball model in Abaqus/CAE is shown in Figure 1. It is worth noting that in many of the previous numerical studies of golf ball impact [3] [4], the dimples are omitted from the golf ball model. While this may not be an issue when the interested quantity is non-local in nature, such as the coefficient of restitution (COR), the role of the dimples cannot be ignored the more complex interaction between the ball and the grooved club faces and it could have a significant effect on behaviors like spin generation.



Figure 1. Abaqus/CAE model of the golf ball

Figure 2 shows the meshed golf ball model. A dense mesh is required on the face of the cover to model the dimple geometry. As a result, the cover is meshed with 159139 tetrahedral elements. The spherical core is meshed with 27269 brick and wedge elements. The size of the model is well within the capability of modern high performance computing.



Figure 2. Meshed Golf Ball Model

In order for the finite element model to become a viable tool for product design, it needs to have the flexibility of adapting to evolving designs. In this study, efforts are put in to make the model parameterized and the model creation process automatic. This is achieved by implementing the model creation process in the Abaqus Scripting Interface with Python. The script that does the model generation has a few design parameters built-in for easy design variation. Figure 3 demonstrates the model's flexibility in handling different dimple patterns and ball construction types.

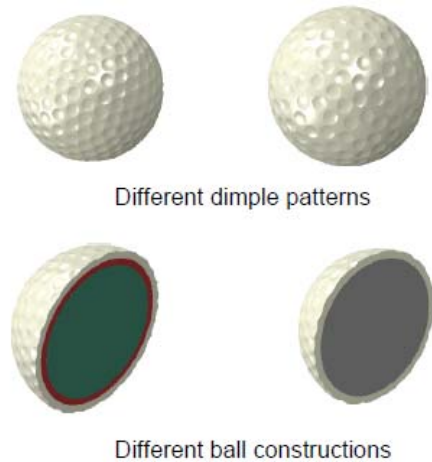


Figure 3. Variations of Golf Ball Geometry Generated via Abaqus Scripting Interface

The golf club modeled in this study follows what is used in the USGA Report On the Study of Spin Generation [5] and is represented by a grooved steel plate. Similar to the modeling of the

ball, the model generating script has the ability to create club plates with different types of grooves from a U-shape to a V-Shape [5] as shown in Figure 4.



Figure 4. Variations of Golf Club Plate Geometry Generated via Abaqus Scripting Interface

3. Material Model

The response of the golf ball during impact is dominated by the more flexible core. As a result, material characterization efforts are mostly dedicated to the core material. For the cover, a straightforward hyperelastic model with generic properties from the literature [3] is used.

The peak strain rate that the core experiences during impact can be as high as 5000 1/sec. Material testing at that strain rate requires special techniques like the Split-Hopkinson Pressure Bar [6]. For this study, the data from Quintavalla and Johnson [7] is used to calibrate the core material model. The stress-strain curve from the uniaxial compression test is shown in Figure 5.

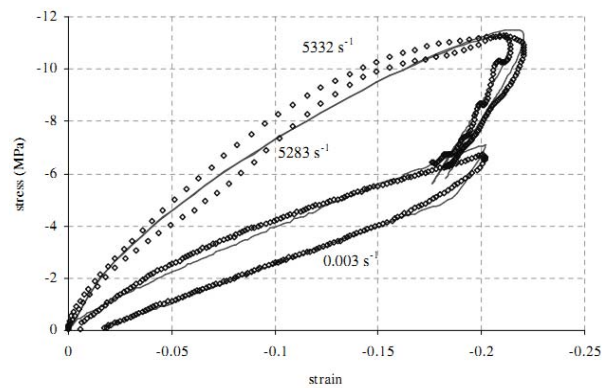


Figure 5. Experimental Data for the Core [7]

Two different material models are calibrated with the experimental data in Figure 5, a simple linear viscoelastic (LVE) material model and an advanced Parallel Network Model (PNM) [8] [9]. The PNM material model is developed by Veryst Engineering and is available as Abaqus User Material Routines (UMAT and VUMAT). These two models are shown in Figure 6 with the experimental curves. We can see that in this case LVE has trouble capturing the hysteresis in the experimental data due to the extreme wide range of strain rates, while PNM does a much better job on that. This key difference will affect the response of the golf ball and will be discussed in the results section.

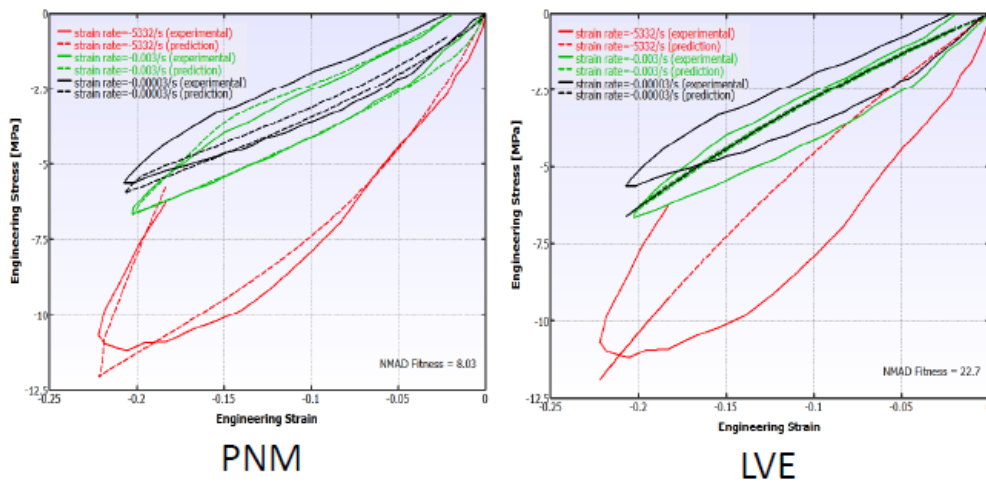


Figure 6. Calibrated Material Models: PNM and LVE

It is also of interest to study the case where high strain rate experimental data is not available. In this case we can only use the experimental curves at lower strain rates to calibrate the material model. Figure 7 shows the calibrated PNM model with the low strain rate experimental curves. The finite element model with this material model will be identified as PNM-low in the results section.

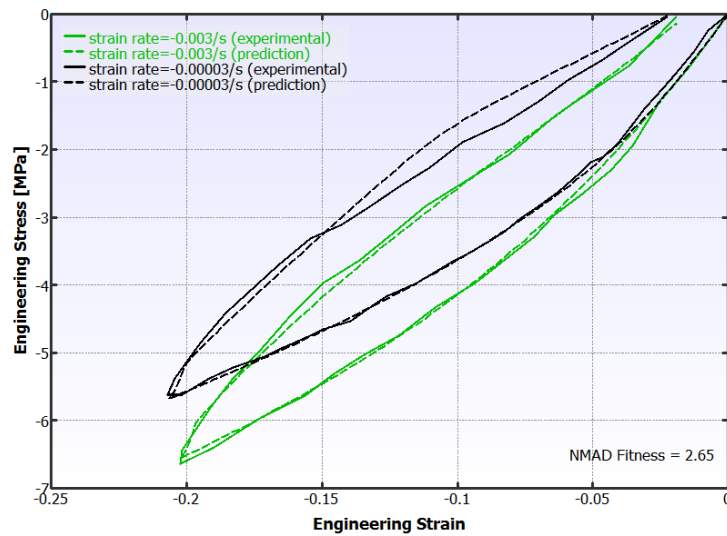


Figure 7. Calibrated PNM Model Based Only On Low Strain Rate Experimental Data

4. Impact simulation

The simulation model of the golf ball impacting the golf club plate is set up in Abaqus/Explicit. A screen shot of the model is shown in Figure 8. Both normal and oblique impact conditions can be simulated but only normal impact with an impacting speed is 34.7m/s is considered for this study. A duration of 2 mili-seconds is simulated. The simulation takes about 2 hours on a 6-core Intel Core i7 workstation with minimal mass scaling.

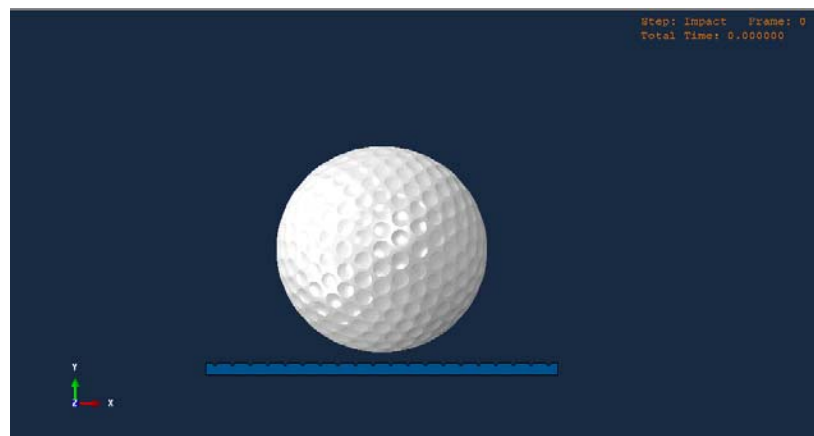


Figure 8. Screenshot of the Abaqus/Explicit Simulation

5. Results

The peak strain rate observed in the simulation is on the order of 4000 1/sec, which is lower than the highest tested strain rate of 5332 1/sec. Figure 9 shows the Mises stress in the core when the ball is compressed the most. The PNM model predicts a peak stress of 14.1MPa while the LVE model predicts a peak stress of 10.4MPa. This difference is due to the fact that the LVE model underestimates the stiffness of the structure in the interested strain range. Figure 10 shows the time-history of the golf ball's kinetic energy during the impact event. We can see that the LVE model predicts very little kinetic energy loss, which is not realistic. This is a consequence of the fact that the LVE model in this case fails to predict the hysteresis of the core material (Figure 6). As a result, the LVE model predicts a higher rebounding height. Figure 11 shows the location of the ball at 2 mili-seconds.

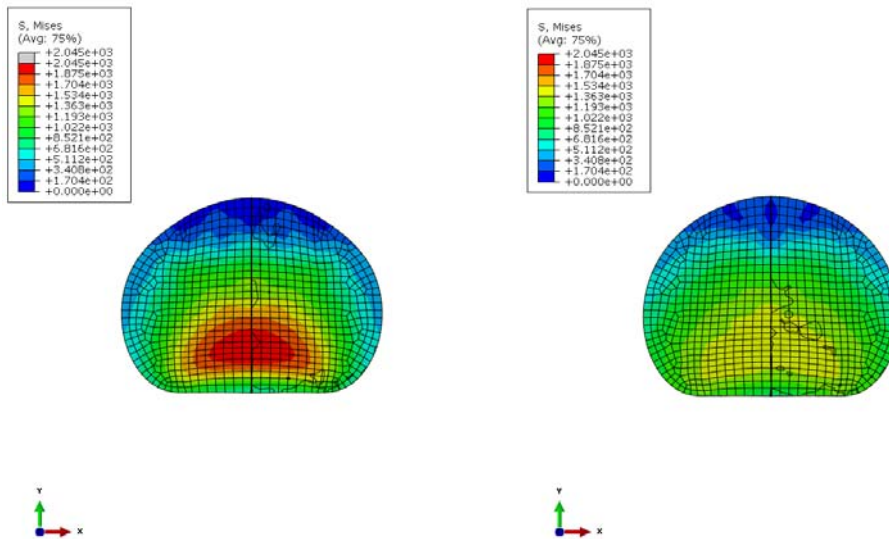


Figure 9. Mises Stress in the Core at Peak Compression

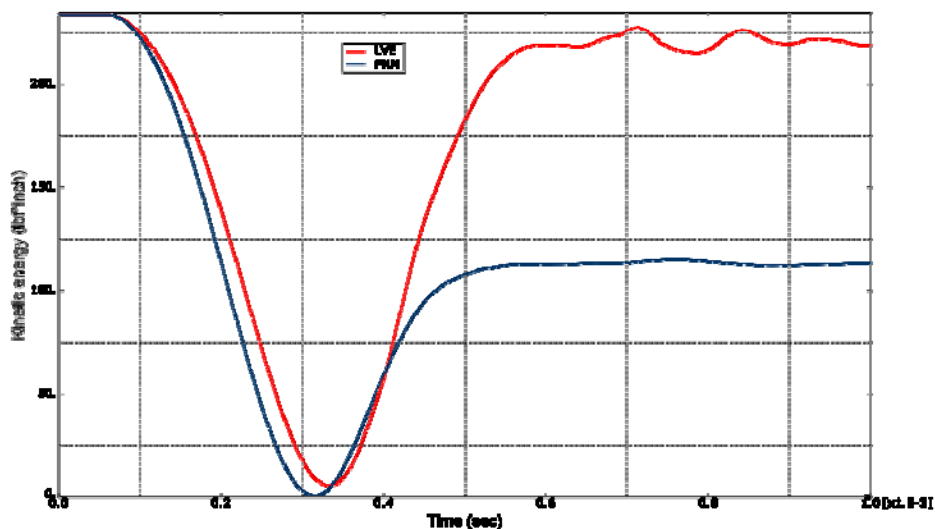


Figure 10. Kinetic Energy Time History Plot, PNM and LVE

It is also of interest to look at the results from the model PNM-low, where only low strain rate experimental data is used for material calibration. The plot of impact force is shown in Figure 11 and we can see that the absence of high strain rate data does cause a considerable change in the impact response of the golf ball.

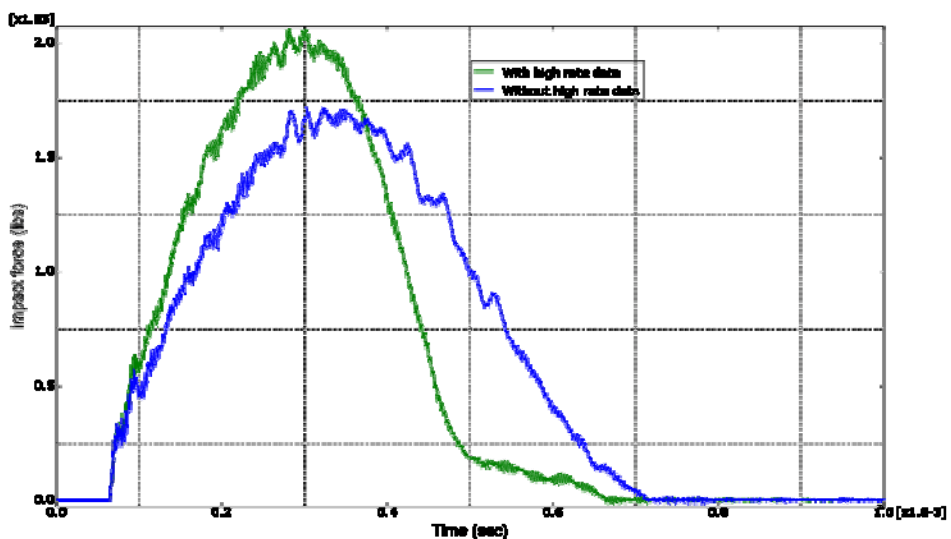


Figure 11. Impact Force Time History Plot, PNM and PNM-low

6. Discussion

The key to the accuracy of the golf ball impact simulation hinges on accurate characterization of the core material. LVE material model works well for events characterized by small strain or by a narrow range of strain rates. Neither condition is true for the golf ball impact event and hence advanced material models like the PNM model has to be used produce trust-worthy results. At the same time, results from this study also show that the knowledge of high strain rate behavior is also very important for correct material characterization and should be pursued.

The design cycles in the golf industry is short and the accuracy requirement for simulations is very high. The work done in this study, highlighted by advanced material modeling and parametric model creation, is capable of meeting the challenge and becomes a powerful design tool.

In-house physical testing has been planned where the impact process will be recorded by a high-speed camera and quantities like impact force will be measured. The data obtained will be used to validate the numerical models described in this work.

7. References

- [1] "Actual Launch Conditions Overall Distance and Symmetry Test Procedure," USGA-TPX 3006, USGA, 2011
- [2] US Patent 5562552
- [3] K. Tanaka, F. Sato, H. OoDaira, Y. Teranishi, F. Sato, S. Ujihashi, "Construction of the Finite-Element Models of Golf Balls and Simulations of Their Collisions," J. Materials: Design and Applications, Vol 220, pp. 13-22, 2006
- [4] J.D. Axe, K. Brown, K.Shannon, "The Vibrational Mode Structure of a Golf Ball," J. Sports Science, Vol. 20, pp. 623-627, 2002
- [5] "Second Report on the Study of Spin Generation," USGA, 2007
- [6] W. Chen, B. Song, "Split Hopkinson (Kolsky) Bar," Springer, 2011
- [7] S.J. Quintavalla, S.H. Johnson, "Extension of the Bergstrom-Boyce Model to High Strain Rates," Rubber Chemistry and Technology, Vol. 77, pp. 972-984, 2004
- [8] The Parallel Network Model (PNM) and the MCalibration software are commercially available from Veryst Engineering (<http://www.veryst.com/PolyUModLibrary.html>).
- [9] J.B. Bergstrom, M.C. Boyce, "Constitutive Modeling of the time-dependent and cyclic loading of elastomers and application to soft biological tissues," Mechanics of Materials, Vol. 33, pp. 523-530, 2001.