Trans. Korean Soc. Mech. Eng. C, Vol. 3, No. 2, pp. 97~106, 2015

<응용기술논문> DOI http://dx.doi.org/10.3795/KSME-C.2015.3.2.097 ISSN 2288-3991(Online)

Need for Accurate Initial Conditions to Simulate Flexible Structures in Motion

Nelson Woo*, Brant Ross*† and Ryan West
* MotionPort, St Georage, UT 84790 USA.,
** The Mistequay Group, Aero Precision Products, Saginaw, MI 48605, USA

(Received February 16, 2015 ; Revised March 31 2015 ; Accepted April 10, 2015)

Key Words: Flexible bodies, Flexible contact, Initial conditions, Multibody dynamics, System simulation, CV joint boot

Abstract: Flexible structures are often important components of mechanical assemblies in motion. A flexible structure sometimes must go through assembly steps that cause it to be in a pre-stressed condition when in the starting position for operation. A virtual prototype of the assembly must also bring the model of the flexible structure into the same pre-stressed condition in order to obtain accurate simulation results. This case study is presented regarding the simulation of a constant velocity joint, with a focus on the flexible boot. The case study demonstrates that careful definition of the initial conditions of the boot and flexible body contacts yields high-fidelity simulation results.

1. Introduction

The practice of multibody dynamics has evolved from rigid body dynamics to the simulation of assemblies that contain a combination of rigid and flexible bodies. Flexible bodies may undergo large deflections and contact other bodies or themselves. These sophisticated models, if created properly, produce results with high fidelity.

Sometimes the flexible structure has a deformed initial configuration due to the assembly process of the product. The simulation model must replicate the initial deformed state of the flexible body in order to obtain accurate results. The challenge is that the starting mesh is in a relaxed state. A series of simulation steps is needed to bring the structure into the correct starting state of deformation. This presupposes that the simulation software is able to save the model, including the deformed structure, at the end of each step such that the model can start each new simulation with the model state from the conclusion of the prior step.

These modeling and simulation concepts will be demonstrated by considering the evaluation of a constant velocity (CV) joint boot. CV joints have been used for a long time, as evidenced by the patent application dated in 1928 that is shown in the left side of Fig. 1. A photo of a typical CV joint is shown in the right side of Fig. 1. The use of CV joints increased substantially with the rising popularity of front-wheel drive automobiles, but they are also used in a variety of all-wheel drive vehicles.

One interesting application of CV joints is with off-highway, all-wheel drive trucks that traverse challenging terrains, as depicted in Fig. 2. In this environment the CV joint can operate with a transmission angle of 40 degrees or more as the steering angle and suspension travel are near their limits. In these conditions the performance of the CV joint boot (see Fig. 3) is very important because it is critical to keep contaminants out of the CV joint. Failures caused by extreme deformation or rubbing on the shaft need to be avoided.

The evaluation of rubbing on the shaft as well as the stress in the boot depends on the dynamics of the operating environment, including the effects of imbalance of the boot. While the boot is well balanced as manufactured, during usage it is possible for a quantity of grease to drop from the CV joint and lodge between the folds of the boot. Given the high rotational speeds of the boot, even the small imbalance caused by the grease can be significant.

Mistequay Group is a designer and manufacturer of custom constant velocity joints for the military, commercial and marine markets. Mistequay desired a simulation tool and process that would allow them to evaluate the behavior of a

† Corresponding Author,

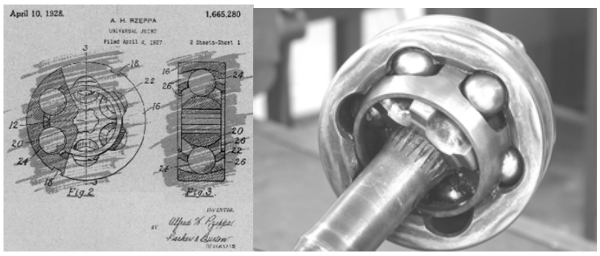


Fig. 1 Patent application and photo of a typical CV joint

Fig. 2 Extreme operating conditions for CV joints

Fig. 3. Boot on a CV joint half shaft

CV joint boot under extreme conditions. One of their customers had requested a half-shaft that could operate at an increased transmission angle. It was important to have a simulation tool that could provide needed insight as to the design of the new boot and that could prevent or reduce costly design iterations. The RecurDyn multibody dynamics software was selected for this project because it had the ability to couple multi-body dynamics with mesh-based nonlinear flexible components[1~6]. It also has the ability to “extract” a model, or use the end conditions of one simulation as the starting point for another simulation. The new model remembers the internal stresses of the flexible structure corresponding to the loaded condition

2. Procedure

The first step was to simulate a model of the existing CV boot design and to compare the simulation results with actual results. This section explains all of the steps needed to prepare the model and to run the series of simulations that are needed to get to the results. The process needed for any particular product assembly varies according the physical operations that are needed to assemble the actual product

The first step is to model the half-shaft assembly in a CAD or PLM tool. Mistequay already had the assembly modeled in NX (see Fig. 4). That model was processed using NX Motion such that the geometry was organized into logical rigid bodies. NX Motion has the ability to export the motion model to RecurDyn. The result is shown in Fig. 5. Only the rigid bodies have been transferred to the model. A rigid body of the CV boot was also transferred and will be replaced with a mesh.

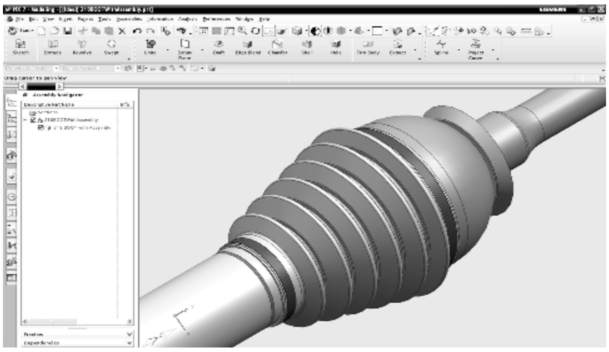


Fig. 4 Boot on the CV Half Shaft as modeled in CAD

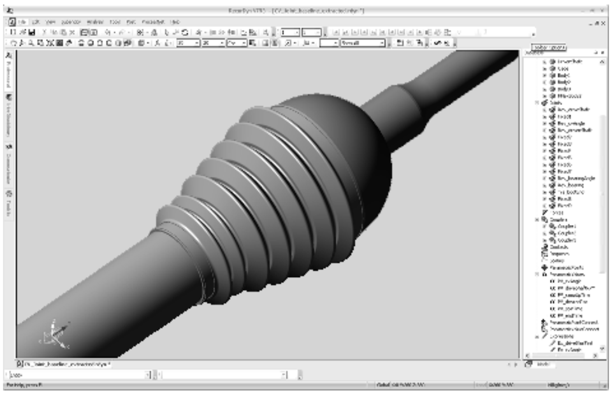


Fig. 5 CV joint model as imported in RecurDyn

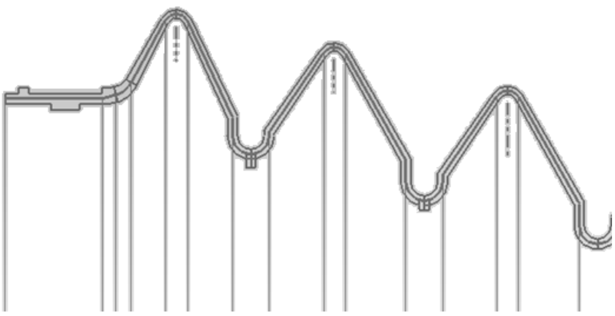


Fig. 6 Profile of the mid-surface of the CV joint boot

The boot geometry in NX could have been used to create a finite element mesh; however that proved to be very difficult because of the varying thickness of the boot along its length. The boot is manufactured with a blow-molding process and a thickness variation is typical.

It may have been possible to generate a mesh with solid elements, but that would have needed to be a mesh with a large number of very small elements. A more efficient approach was to generate a mesh with shell elements in the mid-plane of the solid geometry. Due to the varying thickness of the boot the mid-plane mesh could not be generated automatically. A profile was created from the mid-plane locations provided in the drawing of the boot, as shown in Fig. 6. That profile was revolved to create a surface and a mesh was readily defined on that surface using NX Advanced Simulation, as shown in Fig. 7. Note that the height of the elements in the axial direction was reduced in the regions where most of the bending would take place, namely the tips and roots of the folds of the boot.

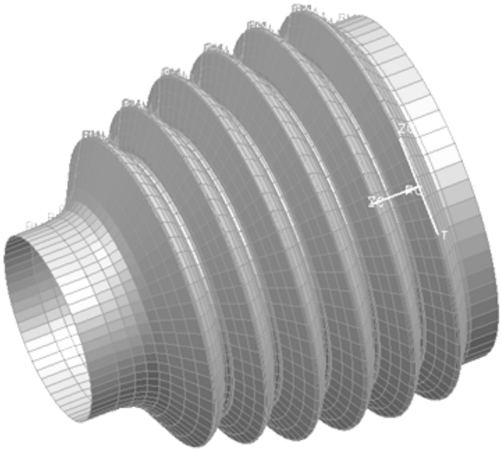


Fig. 7 Boot mesh created on the revolved surface of the boot profile

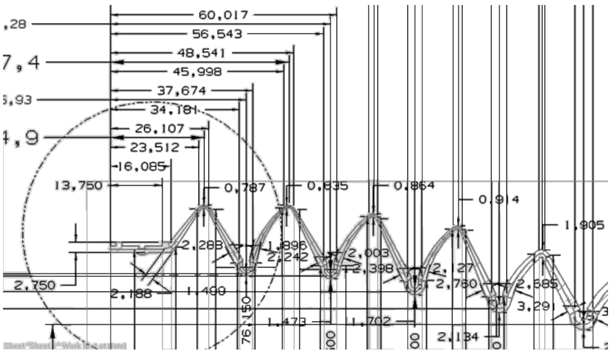
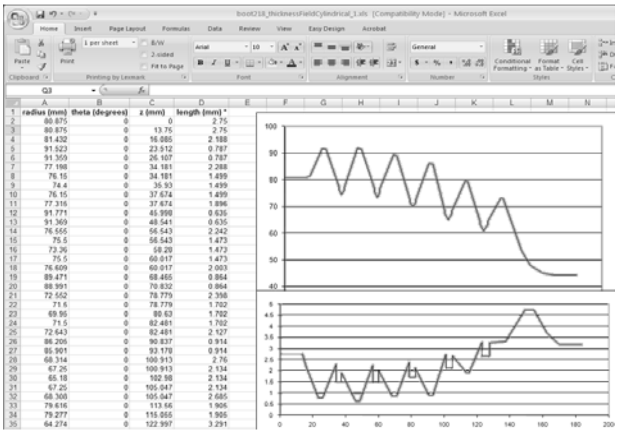


Fig. 8 Boot thickness information from the boot drawing



	radius (mm)	theta (degrees)	z (mm)	length (mm) *
2	80.875	0	0	2.75
3	80.875	0	13.75	2.75
4	81.432	0	16.085	2.188
5	91.523	0	23.512	0.787
6	91.359	0	26.107	0.787
7	77.198	0	34.181	2.288
8	76.15	0	34.181	1.499
9	74.4	0	35.93	1.499
10	76.15	0	37.674	1.499
11	77.316	0	37.674	1.896
12	91.771	0	45.998	0.635
13	91.369	0	48.541	0.635
14	76.565	0	56.543	2.242
15	75.5	0	56.543	1.473
16	73.36	0	58.28	1.473
17	75.5	0	60.017	1.473
18	76.609	0	60.017	2.003
19	89.471	0	68.465	0.864
20	88.991	0	70.832	0.864
21	72.552	0	78.779	2.398
22	71.6	0	78.779	1.702
23	69.95	0	80.63	1.702
24	71.5	0	82.481	1.702
25	72.643	0	82.481	2.127
26	86.205	0	90.837	0.914
27	85.901	0	93.178	0.914
28	68.314	0	100.913	2.76
29	67.25	0	100.913	2.134
30	65.18	0	102.98	2.134
31	67.25	0	105.047	2.134
32	68.308	0	105.047	2.685
33	79.616	0	113.66	1.906
34	79.277	0	115.065	1.906
35	64.274	0	122.997	3.291

Fig. 9 Boot thickness information defined in a spreadsheet

The mesh needed to be modified with the varying thicknesses in order to accurately represent the boot geometry. The boot thicknesses were obtained from the drawing information of the boot shown in Fig 8. That information was placed into a spreadsheet as shown in Fig. 9. The spreadsheet data could be read into NX Advanced Simulation and applied to the mesh. The graphics shown in Fig. 10 provide visual feedback to the user regarding the thickness of the various elements.

Now that the mesh is prepared it is written out in a Nastran bulk data file format and then imported into RecurDyn as the boot flexible body. At this point the mesh is in a relaxed state. A set of nodes is defined at the top of the boot and another set of nodes is defined at the bottom of the boot. Boundary conditions are defined with these node sets to constrain the nodes to be fixed in the axial direction of the boot.

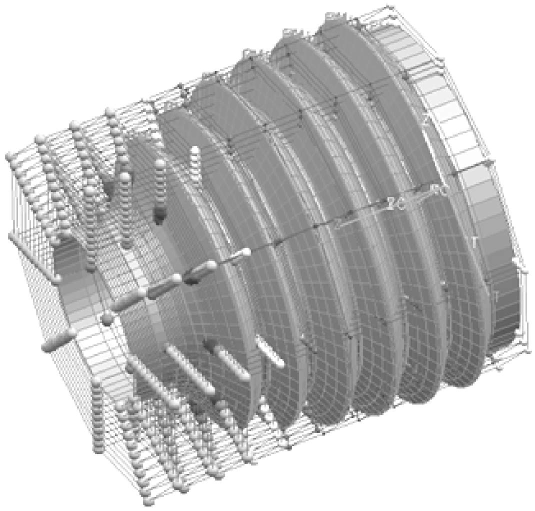


Fig. 10 Graphical feedback on the boot thickness

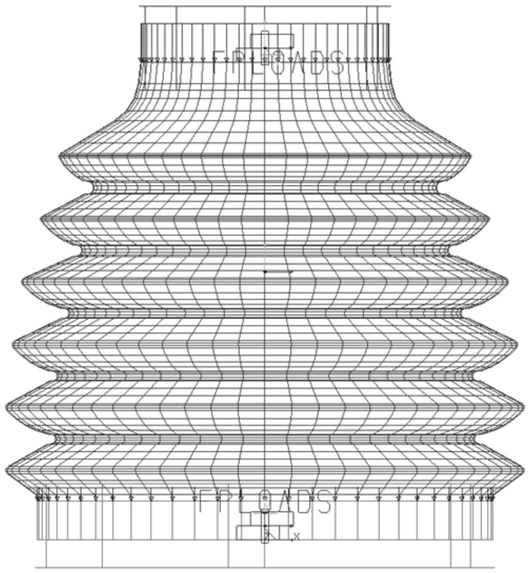


Fig. 11 FE Modeling Approach for Boot

Preparations are made for the flex body contact between the boot and the shaft of the CV joint. The wide end of the boot will contact the bell housing of the CV joint. The narrow end of the boot will contact the shaft. The narrow end of the boot is larger than the shaft; therefore the elements need to be pressed inward until they make contact with the shaft. This effect is similar to that of a hose clamp. The wide end of the boot is not as large as the bell housing. Therefore pressure needs to be applied to the wide end to stretch the elements, then the bell housing is inserted within the wide end of the boot, and then the pressure is released. The contact between the element faces and the bell house allows the boot end to form itself on the bell. The contacts, boundary conditions and pressure loads are shown in Fig. 11.

For each contact a group of element faces is defined (known as a patch set). In the case of shell elements the elements are simply placed in a group. The selection of the front or back face is made automatically according to the assigned direction of the contact on the boot.

Two simulations are run (see Fig. 12) in order to connect the boot to the input and output shafts. As shown on the left the loads in the first model are to compress the narrow end and expand the wide end. After the first simulation the model is extracted and the nodes at the narrow end of the boot are locked in place with a rigid element (similar to the Nastran RBE2 element, as shown in Fig. 13). The connections between the master node and the slave nodes for each rigid element are shown with lighter vectors in the figure. The flexible body contact between the wide portion of the boot and the bell housing is turned on. The second simulation is run and the wide end of the boot will form around the bell housing, with its contraction limited by the contact with the bell housing. A new model is extracted and the nodes at the wide end of the boot are locked in place with a rigid element, again as shown in Fig. 13. The advantage of locking the nodes into place is that the forces and contacts can be turned off in order to speed up the simulation. Once the boot is properly mounted the focus switches to the deformation of the folds and the overall deformed shape of the boot.

The last step to preload the boot is to compress the boot axially to its installed position, as shown in Fig. 14. The wide end of the boot is translated along the flat surface of the bell housing. The boot is now properly preloaded.

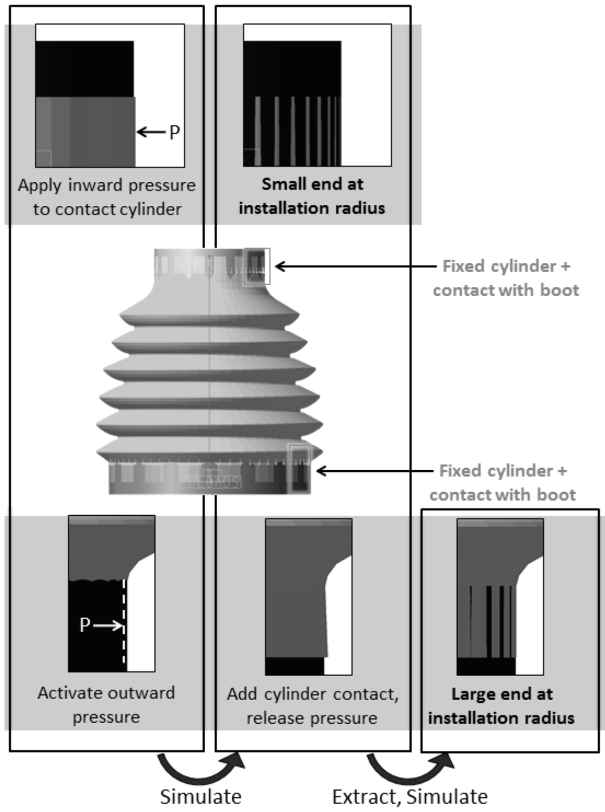


Fig. 12 Sequence of Simulations to pre-load the boot

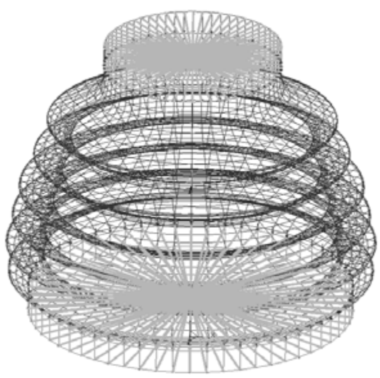


Fig. 13 Freezing the boot ends in the formed configuration

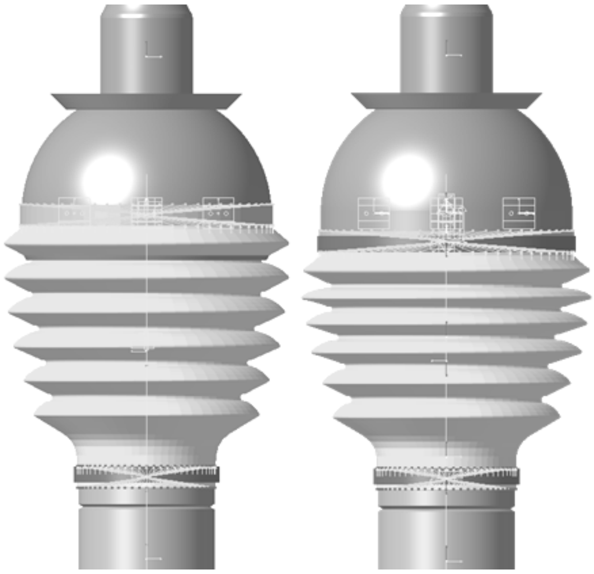


Fig. 14 Axial compression of the boot

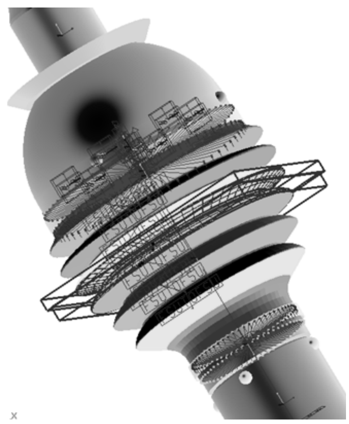


Fig. 15 Defining contacts between folds

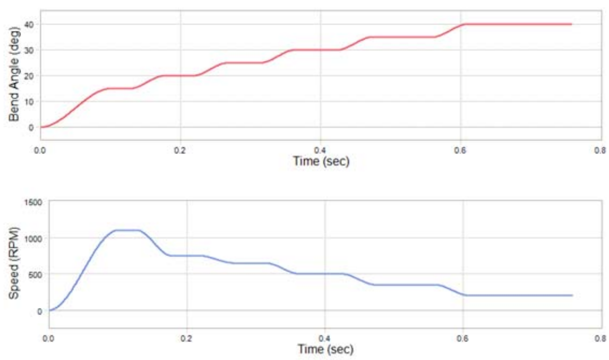


Fig. 16 Running the simulation through angles of interest

Contacts between each pair of fold surfaces are defined as shown in Fig. 15. A patch set of element faces are defined for the top and for the bottom of each fold. A set of contacts are defined for each combination of an upper surface and a lower surface. The model is ready for a dynamic simulation.

The dynamic simulation is designed to evaluate a sequence of combinations of increasing transmission angle and decreasing rotational velocity, as shown in Fig. 16. The logic of this arrangement is that the transmission angle is increased with a higher degree of steering as well as by the traversal of extreme terrains. It is reasonable to expect that the driver will slow down during extreme steering as well as extreme terrains.

As can be seen the transmission angle ramps to 15 degrees in 0.1 sec, then increments the transmission angle 5 degrees at a time. The time intervals are planned such that the model rotates a similar amount at each transmission angle.

The result of the dynamic simulation can be expressed as animations and plots of data. For purposes of validation in this study there is a focus on two comparisons between the simulation results and physical testing.

First, when the production boot is rotating with a transmission angle that exceeds the design specification, the outer edges of the top two folds collapse at the outer portion of the rotation, forming dimples. Fig. 17 shows the excellent correlation

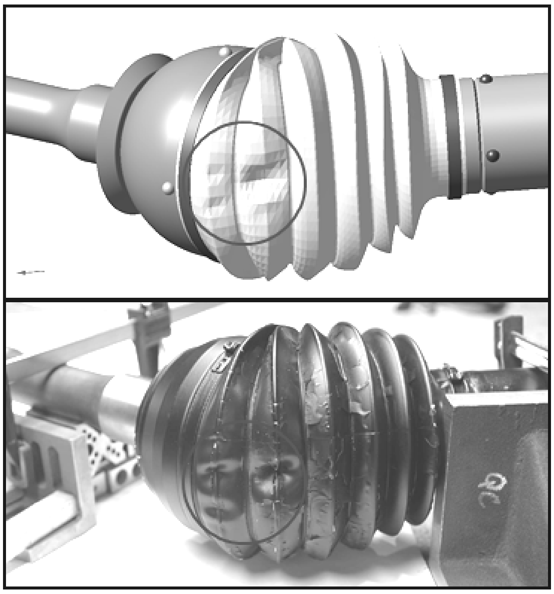


Fig. 17 Comparison of dimple formation in simulation and test

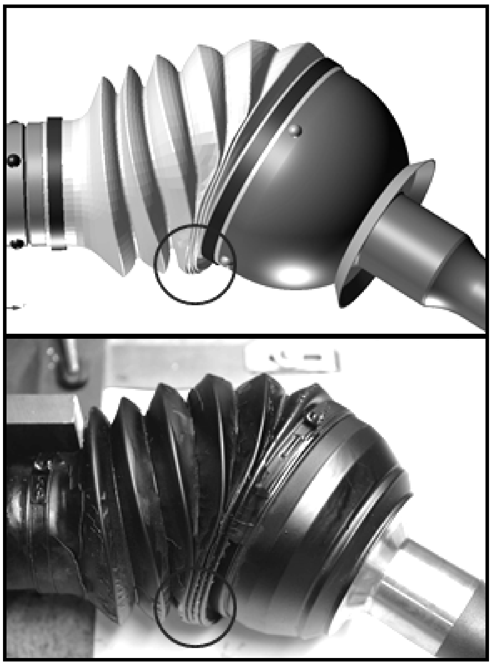


Fig. 18 Comparison of boot fold contact in simulation and test.

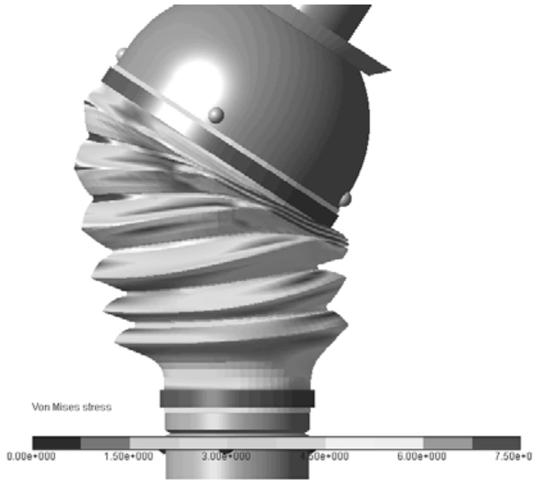


Fig. 19 Stress contours just before the onset of dimple formation

between the simulation results (top) and the photograph from physical testing (bottom). The locations of the dimpling are circled. The dynamics of this situation are more apparent when watching the animation from the simulation because it can be seen that the dimpling effect is under constant transition. Depending on the transmission angle, there can be transitions between 1 and 2 dimples on a fold or an oscillating behavior between the two folds. Observers of the physical testing under these extreme conditions report that during dimple formation they actually hear popping noises from the boot.

3. Results

The second comparison considers the shape of the boot and the contact between folds. Fig. 18 shows excellent correlation between simulation and test. A consideration of shape is critical because a design requirement is that the deformed boot must not contact the underlying shaft since rubbing can cause premature failure. The animation of the plane view of the CV joint can be used to quickly check for possible interference.

Given the excellent validation shown in the figures there can be high confidence in the other outputs of the simulation, including stresses and strains in the boot, such as is shown in Fig. 19. Simulation outputs can provide valuable insights. For example it is interesting to see the evolution of the stress. 19. Simulation outputs can provide valuable insights. For example it is interesting to see the evolution of the stress in the boot as the transmission angle of the joint gradually increases.

4. Benefit to Design

The benefits of using simulation include:

- Reduced number of iterations. Reworks of a tool are US$5-10K and require 3-4 weeks of time. New boot tools are approximately a US$40K investment and require 12-16 weeks of lead time.
- Having accurate boot fold positions from simulation helps in the design of proper clearances to the bar shaft under the boot along with generating the external operating cone of the boot, which ensures clearance at jounce and rebound conditions during chassis kinematics.
- Simulation further helps design the boot to operate at the angle and plunge required, meaning the determination of the correct length and height of the folds to ensure there is enough motion from the boot to meet the angle of the joint. This helps eliminate "puckering" (overstretching) of the folds at angle.
- Simulation enables the evaluation of what happens to the boot once an eccentric mass of grease is added. Is it going to fail or contact the bar?

5. Conclusions

The following can be concluded from this study:

- Today's simulation software provides capabilities to accurately model assemblies of rigid and flexible bodies in motion, including contact.
- Accurate behavior of flexible bodies requires the careful replication of initial, pre-loaded conditions.
- Substantial savings in cost and time result from the ability to simulated complex assemblies and structures such as can be found in a CV joint.

References

(1) 2014, RecurDyn Help Manual, FunctionBay Inc., Seoul, Korea.

(2) Cho, H. J., et. al., 2001, "A Recursive Implementation Method with Implicit Integrator for Multibody Dynamics," *Proceedings of ASME Design Engineering Technology Conference*, Third Symposium on Multibody Dynamics and Vibration, September 9-12, Pittsburgh, PA, USA

(3) Bae, D. S., 2005, "Development of a New Multi-Flexible Body Dynamics (MFBD) Platform: A Relative Nodal Displacement Method for Finite Element Analysis," *Proceedings of the ASME 2005 IDETC & CIE Conference*, September 24-28, Long Beach, California USA, DETC2005-84723.

(4) Ross, B. A., 2005, "History, Perspective and Outlook for Media Transport Simulation Using Multibody Dynamics," *Proceedings of the ASME IDETC 2005 Conference*, ASME Fifth Symposium on Multibody Dynamics and Vibration, September 24-28, Long Beach, CA, DETC2005-85191.

(5) 2009, "Early Experiences in Using Multibody Dynamics Software to Simulate Flexible Webs in Production," *Proceedings of the ASME IDETC 2009 Conference*, ASME Seventh Symposium on Multibody Dynamics and Vibration, August 30 - September 2, San Diego, California, USA, DETC2009-86719

(6) Choi, J. and Choi, J. H., 2013, "A Smooth Contact Algorithm Using Cubic Spline Surface Interpolation for Rigid and Flexible Bodies," *Proceedings of the ASME IDETC 2013 Conference*, ASME Ninth Symposium on Multibody Dynamics and Vibration, August 4-7, Portland, Oregon, USA, DETC2013-12117.

Brant Ross has a Ph.D. degree in Mechanical Engineering from Brigham Young University and has spent over 25 years working in multibody dynamics. He is a registered professional engineer in Indiana since 1987. He worked for Mechanical Dynamics for 9 years and founded MotionPort 11 years ago. He has given 38 presentations at professional meetings, published 7 papers in refereed publications and has published 1 journal article.

Nelson Woo has been working as an Application Engineer in the area of Multibody Dynamic Simulation. He has experience utilizing this technology in a wide range of applications including biomechanical research, space satellites, military and agricultural vehicles, media transport, and the automotive industry. He previously graduated with a Master's degree in Mechanical Engineering from the University of Michigan. He also has professional experience as a software developer, after receiving a Bachelor's degree in Computer Engineering from the University of Michigan.

Ryan West is a Sr. Design Engineer at Mistequay Group in Saginaw, Michigan. Mistequay Group is a designer and manufacturer of custom constant velocity joints for the military, commercial and marine marketplace. Ryan has been at Mistequay Group for 6 years, prior to that he has worked in the areas of automotive steering column design and aerospace ballscrew design for commercial and military flight control applications. Ryan is a graduate of Saginaw Valley State University in Saginaw, Michigan.