Modeling of a Tactical Spike Strip for use at Checkpoints

Stephen S. Recchia, David Geissler, Matthew Sabolcik, and Martin Roswall

U.S. Army – ARDEC, Picatinny Arsenal, Picatinny, NJ 07806

Dassault Systemes Simulia Corp., 166 Valley Street, Providence, RI 02909

Abstract:
It is sometimes necessary to strategically place temporary checkpoints in theatre. One such tool for creating an effective checkpoint is a deployable spike strip. One fault of the current system is that puncturing and deflating the tires may not be sufficient to bring the car to a halt. The U.S. Army proposed a request for improvement to design a new spike strip that actually causes the vehicle to come to a stop once the deployed system is rolled over. Tests of this new system are both costly to run and time consuming. A finite element simulation of the proposed system was created in Abaqus/Explicit to provide insight into how the system operated. Once set up, the model could then be used to down-select between subsequent iterations. Comparison with testing showed that the Abaqus/Explicit simulation could predict the pullout force between the spike and the tire to a high level of accuracy. Once the pullout force was validated, the interaction between the fabric strips connecting the individual spikes and the underside of a vehicle was investigated. The analysis showed that simple nylon fabric strips were too weak to stop the vehicle's tires from spinning. Changing to Kevlar strips resulted in the spike being pulled out of the tire prematurely. As a result of this analysis considerable time and cost was saved and currently alternative designs are being considered.

Keywords: Abaqus Explicit, Kevlar, Fabric, dynamic, tire, Spike Strip, non-linear

Table of Contents

Introduction....................4
Spike Strip Operation....................4
Obtaining 3D Model Geometry....................5
Experimental Tests....................6
Finite Element Analysis....................7
FEA Validation of Spike Pull Removal Test....................7
Simulating Kevlar with Fabric Material Model....................9
Large Scale Analysis....................10
Conclusions....................11
References....................12

Table of Figures

Figure 1 - Ford Taurus model provided to ARDEC....................6
Figure 2 - Experimental setup of spike pull test....................6
Figure 3 - Material sections....................8
Figure 4 - FEA analysis of spike pull out test....................8
Figure 5 - Plain-Woven Kevlar Fabric under Quasistatic Uniaxial Tension....................9
Figure 6 – Kevlar membrane collapsing onto a sphere....................9
Figure 7 - Large Analysis Assembly....................10
Figure 8 -Spike strip analysis result....................11

Acknowledgements

Without Dave Haberman and Abaqus East acting as a resource to bring the Abaqus simulation community together this analysis would never have been completed. Many thanks go to the excellent support we have received from them.

Introduction

Soldiers would like to be able to quickly deploy a vehicle-stopping device in the event that someone tries to drive through the checkpoint. The device must be able to stop a vehicle in such a way that results in minimal harm to the passengers and minimal damage to the vehicle. Current portable designs for vehicle stopping devices require soldiers to physically move the mechanism in order to achieve on-off-on capability; the desire is to have a system that is man-portable, able to be set up within minutes, and can be automatically deployed or retracted across a road. One of the proposed designs is lighter and more easily deployable, but for it to be successful some engineering parameters need to be optimized. In order to save the program the cost of iterative testing, a finite element approach using Abaqus Explicit was implemented.

This report describes how a finite element analysis was set-up and validated for the baseline of one proposed design. Once an analysis of the baseline has been created it can be used to optimize performance parameters such as spike or fabric material choice.

In order to accurately capture the physics of the system, a model of a tire had to be created and a detailed model of the underside of a vehicle needed to be obtained.

Spike Strip Operation

The spike strip must be designed to stop a wheeled vehicle weighing up to 5,500 lbs traveling at speeds up to 30 mph within 200 feet on dry paved or soft surface roadways. When a vehicle rolls over the strip, spikes puncture the tire and wrap flexible straps of material around the axle to stop the vehicle from moving. The purpose of the spikes is not to deflate the tire. Their purpose is to initiate the straps wrapping around the axle. Once the fabric has entangled the axle, it impedes further axle rotation keeping the vehicle in place until the fabric is removed from the vehicle. Once an optimum strip design is achieved, the desire is to then automate the system in such a way that it deploys and retracts across a road without a soldier having to physically move the strip and put himself in harms way.

Obtaining 3D Model Geometry

The spike strip assembly was built in house using the ProE Wildfire 4 3D modeling code. This geometry was brought into Abaqus explicit using the Elysium translator.

The tire was derived from an Abaqus Explicit example Answer ID 3007 “How do I create a treaded tire model using the symmetric model generation capability and then use the model in a steady state transport analysis?” This problem was completed in four parts and resulted in a tire geometry that could not be edited. In order to have a part that could be modified, a sketch of the tire profile was created and revolved into a 3D part. The tread of the tire was of less importance to this analysis than the average thickness and rebar placement.

A 2001 Ford Taurus (Fig. 1) was chosen as the vehicle for this analysis because it is a close representation of the type of vehicle most commonly used in areas where this spike strip is likely to be deployed. This model was originally downloaded from National Crash Analysis Center (NCAC).
The Federal Highway Administration (FHWA) and National Highway Traffic Safety Administration (NHTSA) through the NCAC have put considerable resources into the development and validation of the posted finite element model. These finite element models are available over the world wide web at no cost from the NCAC. The original model from NCAC was translated from LS DYNA to Abaqus. It was this Abaqus model (.inp) that was the starting point of the car geometry for this project.

SMULIA has used this model as a base for demonstrating their automotive Unified FEA concept. The Unified FEA concept consists of structural crashworthiness analyses (USNCAP Frontal Impact, IIHS Side impact etc.), NVH (Noise, Vibration and Harshness) analyses (Eigenmodes, Mobility etc.), and system level durability analyses such as pothole impact, curb strike, and rough road (performed using the co-simulation technique available between Abaqus/Standard and Abaqus/Explicit).

Figure 1 - Ford Taurus model provided to ARDEC

Experimental Tests

The first question any good analyst should ask is, “Is my model accurate?” The answer lies in how closely the models prediction represents empirical results. One area of concern was the 3D model of the tire and the force required to pull a spike out of it. For this reason it was decided that before a full analysis was created, the spike removal force would be validated against experimental data. A series of tests were set up where a rigid spike strip was pulled from the tire (Fig. 2).

Figure 2 - Experimental setup of spike pull test

After these simple tests were conducted, it was determined that the average removal force for one spike was 2,500 Newtons.

Finite Element Analysis

The finite element analysis of the proposed baseline design was broken into three parts. The first part of the analysis was to validate the predicted spike removal force to empirical testing data. In order to get an accurate result, the tire needed to be modeled in some detail. The overall shape was created, and then sections were partitioned to represent the different layers in an actual tire. The second part was to ensure that the correct element was chosen from the fabric straps. It is very important that the element used was not too stiff in the bending direction. The third part of creating a baseline analysis was to build a large scale 3D model of the dynamics, including the vehicle, tire, and spike strip system.

FEA Validation of Spike Pull Removal Test

The first part of the analysis was to model the force required to pull out a spike that has already punctured the tire. When building the 3D model, the overall shape was created and then sections were partitioned to represent the different layers in an actual tire. The reinforcing belts were modeled as skins. There were eleven sections defined in this model (fig. 3).

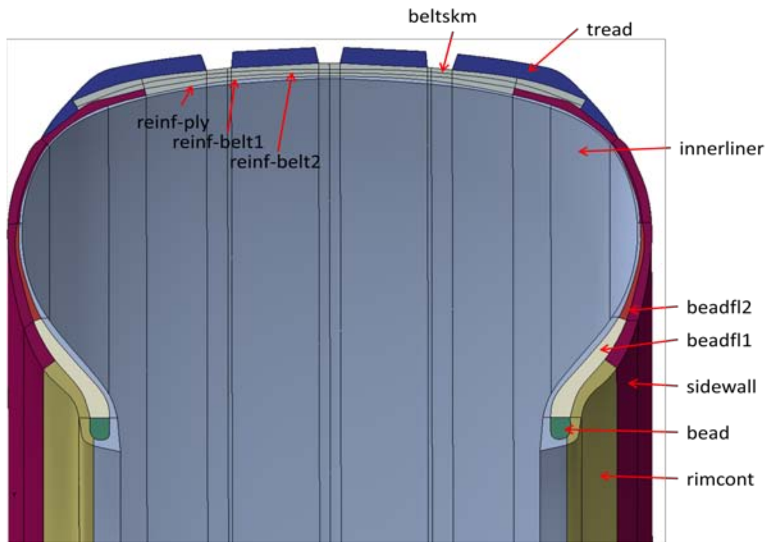

Figure 3 - Material sections

Once the tire model was built, a finite element model was created with one spike embedded in the tire. That spike was then displaced 80mm over 10 ms. The force required to meet that displacement was recorded and plotted against time (Fig. 4).

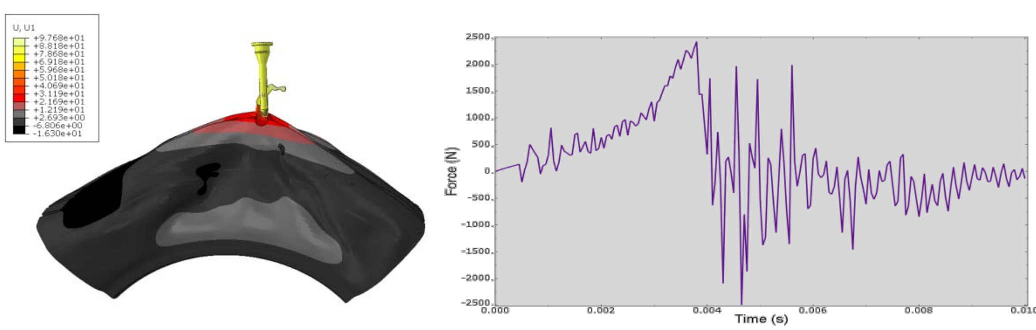

Figure 4 - FEA analysis of spike pull out test

Testing determined that 2,500 N was required to remove a spike from the tire. The analysis showed that the peak force to pull out the spike was 2,489 N. It was decided that given the assumptions made to model the tire material and the inaccuracies during testing these values were in close enough agreement to move onto building a larger scale model.

Simulating Kevlar with Fabric Material Model

The next hardest phenomena that was inherent in building the finite element analysis was modeling the correct response of the fabric strap interacting with the underside of a vehicle. The material parameters were derived from research conducted at Army Research Labs in March 2005 [2].

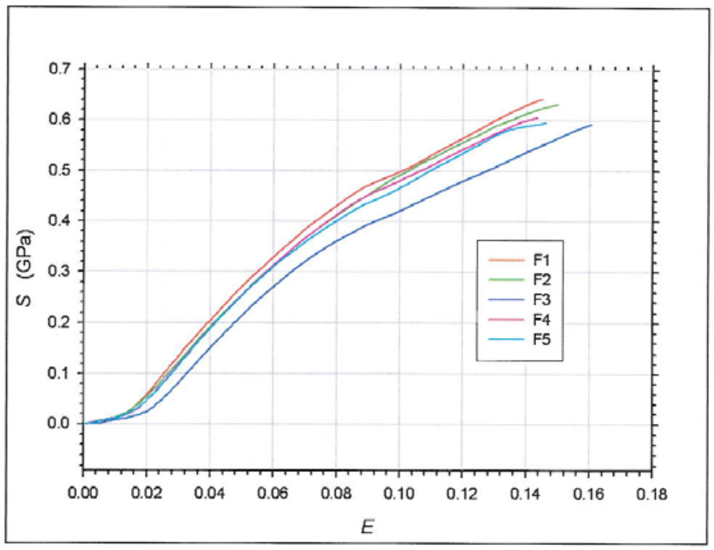

Figure 5 - Plain-Woven Kevlar Fabric under Quasistatic Uniaxial Tension

When modeling a fabric, the ideal reaction would be for the material to not resist any bending. This implies that the correct element, whose formulation fits the physics that is being modeled, must be selected. Shell elements were attempted, but their overall response was too rigid. Beam elements were also tried, but they were also too stiff of a response. The method that seemed to work the best was modeling the fabric as a membrane using the *FABRIC constitutive model built into Abaqus Explicit.

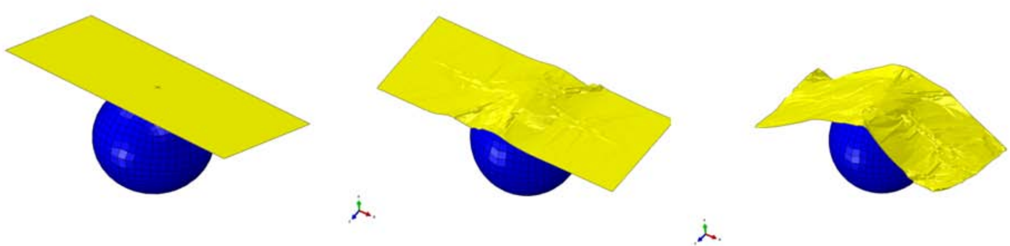

Figure 6 – Kevlar membrane collapsing onto a sphere

Large Scale Analysis

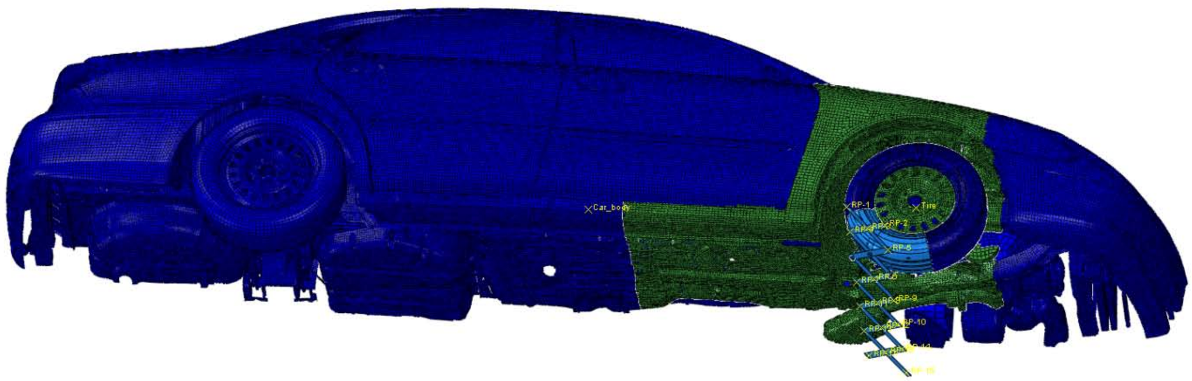

Figure 7 - Large Analysis Assembly

The object of this analysis was to see if the spike strip system would be strong enough to stop the vehicle. The vehicle was given an initial forward velocity of 15 mph with the tire rotating appropriately. The following assumptions were made about the boundary conditions of the problem. No ground was created in the assembly and no gravity was applied in the model. The vehicle was made rigid. The correct mass of the hub was applied through an inertia node coupled to the inside rim of the tire. The wheel assembly rotation was applied to the tire and the representative inertial node. For this run internal tire air pressure was neglected. Each metal spike was coupled to the fabric strip with a kinematic coupling constrain held in six degrees of freedom.

As the tire rotates the middle section of the spike strip hits the undercarriage of the car (Fig. 8). This induces the spike to start pulling on the tire. This design in its current form pulls the spike out of the tire before the car is able to fully decelerate.

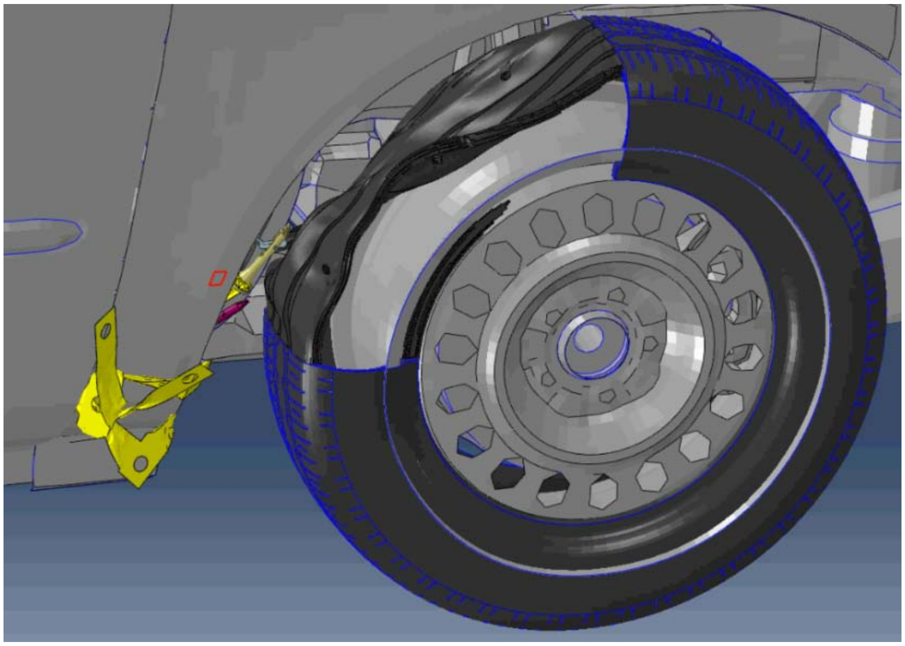

Figure 8 -Spike strip analysis result

Conclusions

For successful operation, the entire system has to dynamically respond to the external loading as predicted. In its current state the spike strip does not react as planned. There is a good level of confidence in the results due to the validation completed on the subsystem levels. One of the benefits of running an FEA is the ability to diagnose the system after unexpected results are produced. As a result the FEA has shown some weaknesses inherent in the baseline design:

- The spike needs to be redesigned to have a larger removal force,
- The stresses seen in the fabric showed marginal survival due to the non-constant stress being applied.

The next iteration will include a stronger spike design to increase the fore required to pull it out of the tire. It will also include a method to elongate the straps to facilitate a constant stress on the fabric once a maximum value is reached.

References

1. ABAQUS Inc., (2010), “ABAQUS User Manual V6.10.”
2. “Plain-Woven, 600-Denier Kevlar KM2 Fabric Under Quasistatic, Uniaxial Tension” M. Raftenberg, M. Scheidler, T. Moynihan, C. Smith, Army Research Laboratory, Aberdeen Proving Ground, MD 21005-5066, ARL-TR-3437, March 2005
3. National Crash Analysis Center , The George Washington University's, Virginia Campus, http://www.ncac.gwu.edu/vml/models.html