19th International Symposium of Ballistics, 7–11 May 2001, Interlaken, Switzerland

METHODOLOGY FOR HARDENING ELECTRONIC COMPONENTS FOR GUN LAUNCH SURVIVAL

M. Berman[1], S. Wilkerson[2], D. Hopkins[2], G. Gazonas[2], A. Frydman[1], D. Carlucci[3]

[1] *US Army Research Laboratory, 2800 Powder Mill Road, Adelphi, Maryland 20783*
[2] *US Army Research Laboratory, Aberdeen Proving Ground, Maryland 21005*
[3] *Armament Research Development and Engineering Center, Picatinny Arsenal, New Jersey 07806*

The Army Research Laboratory (ARL) has developed a comprehensive modeling strategy to support a new SADARM design. The models include fully transient three-dimensional models of SADARM components. The models utilize material properties, component characterizations and component failure criteria experimentally measured and developed by ARL. The models consist of up to 500,000 elements and are used to simulate the complex interactions between the SADARM submunition, carrier and gun barrel. Several post-processing techniques were developed to support the analysis of these models. ARL, the SADARM project office and the prime contractor are utilizing ARL's modeling expertise to maximize the margin of safety of the new SADARM design. This paper describes the simulation methodologies that are being used to drive the design of the new SADARM submunition.

INTRODUCTION

The US Army is currently developing a new class of autonomous "smart" munitions. The new munitions differ from their predecessors through the use of onboard sensing, guidance and control components. Previous Army projectiles were simply aimed at the target and fired, obtaining lethal end results through area explosive effects. The effectiveness of these projectiles depended on the selection of the aim point, the accuracy of the shot and the ability of the platform ballistic software to compensate for environmental conditions. This new class of munitions contains on-board sensors, actuators and intelligence that provide guidance, control, sensing and terminal targeting. As a result a single "smart" projectile effectively replaces several conventional projectiles-providing a variety of benefits to the soldier.

The projectiles are more costly, and in order to carry out their mission, the on-board systems in the new munitions must be packaged to survive the gun-launch environment. In addition, the current emphasis on commercial off-the-shelf (COTS) technology means that many of the electronic components in these systems have not been designed with the gun-launch environment in mind. These issues mandate that careful attention be paid to the structural design to ensure that the munitions reach their targets intact and capable of functioning as intended.

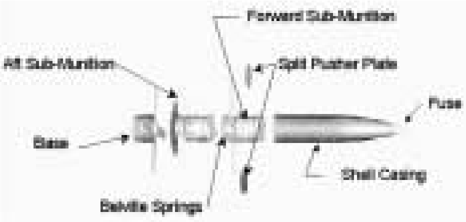

Figure 1. Sadarm projectile.

The SADARM (Sense and Destroy Armor) projectile consists of a base unit, shell casing, fuse, split pusher plate, spring and two lethal mechanisms/submunitions (Fig. 1). The current SADARM design includes a potted electronics module assembly (EMA). This assembly contains most of the electronic circuitry for the submunition. The new EMA design will contain three circuit cards and both potted and unpotted designs are under consideration. These internal components are sensitive to off-axis loading such as that caused by spin rate or the rapid unloading of the projectile's base that occurs at muzzle exit. Additionally, stress waves caused by balloting and impedance mismatching within the stack design serve to further reduce the design margin. The Army Research Laboratory (ARL) is working with the US Army Tank-Automotive Armaments Command-Armament Research Development and Engineering Center (TACOM-ARDEC) and SADARM contractors to develop models and simulations to compensate for the reduced design margin and the increased source of potentially damaging loads. The techniques presented here represent a modern approach to analyzing the projectile's launch loading conditions compared to traditional techniques.

The authors of this paper wish to thank the Project Manager-Artillery Munitions Systems (PM-ARMS) for providing the funding to carry out this research as well as the ARL Major Shared Resource Center and the Department of Defense High Performance Computing Modernization Office for providing the computational resources necessary for the SADARM simulations.

STACK MODEL DETAILS

A complete finite element model of the SADARM projectile and gun system has been built to examine the launch and shot-exit transient loads on the projectile's major structural components. This model can also be used to perform the detailed analysis of individual components during launch as well.

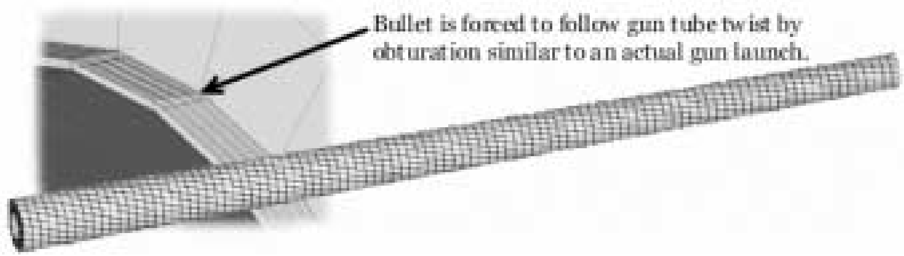

Figure 2. Gun-tube finite element model and gun-projectile interface.

Home
Table of Contents

For this analysis the M199 gun tube is modeled using 8 noded linear brick elements (Fig. 2). A linear twist of 20 to 1 calibers is included in the model; this is equivalent to the actual twist in the 155 mm system.

The rear of the projectile is subjected to a pressure loading from the propellant. Using the IBHVG2 interior ballistics code, predictions are made regarding the launch of the SADARM projectile. The interface between the obturator and the tube was modeled with DYNA3D sliding surfaces. A traction force was added to the bullet's base at the approximate location of the obturation equivalent to the force of the friction as calculated by the IBHVG2 interior ballistics code. This prevented excessive deformation in the obturator while still allowing the bullet to spin up at the proper rate. The pressure load was balanced with the actual friction so that the projectile exited the gun tube at the correct velocity and spin rate.

Both the 8S and 7R base loading conditions were considered. Observed failures have been at higher load rates and in the aft submunition. The shell casing was also modeled using 8 noded brick elements. The base of the shell housing is rigidly attached to the base along the common interface; sliding interfaces were used along the inner wall between the housing and the base. This combination results in a reasonable representation of the actual one-and-one-half thread attachment of the base to the case.

The fuse components were broken down into lumped mass equivalents. Details in any of the models internal components can be added to examine one portion of the flight body at any time. However, this initial model is only being used to examine the load-carrying structural members.

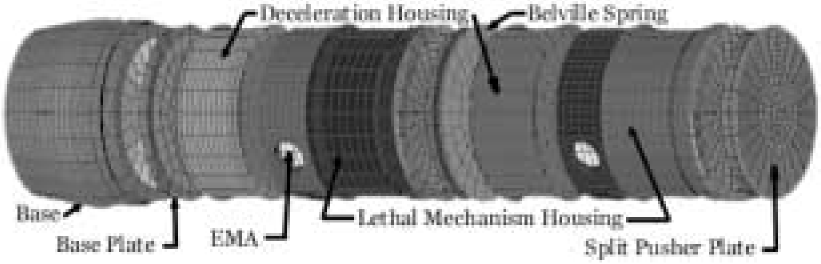

Figure 3. Finite element model of Sadarm's stack components.

The stack model consists of a number of components, all modeled with 8 noded brick elements (Fig. 3). The model's primary use is to examine the load distribution and structural aspects of the internal SADARM stack assembly.

Details can be added to any particular component and to examine individual parts. The parachute and other components inside of the deceleration housing are modeled as an equivalent lumped mass. The lethal mechanism contains several components also modeled as equivalent lumped masses. Inside the lethal mechanism the EMA assembly and the Belleville spring are treated as lumped masses. The Millimeter Wave (MMW) assembly's components are also modeled using an equivalent mass. Sliding interfaces were included between parts.

During the assembly of the SADARM stack, a hydraulic press is used to compress the components and Belville spring. To simulate the compression of the spring in the model, two plates are connected in the middle and pressurized along their inner surfaces. Initially a "no-gap" condition is assumed between the Bellville springs two surfaces. It is believed that a gap exists between the plates, but the gap is not addressed in this model.

The primary purpose of this model was to examine the load-carrying portions of the SADARM projectile. The model provides an accurate method for simulating the transient launch environment of a 155 mm projectile. Furthermore, this model can be used as a flight vehicle to examine individual components. For example, a model of the EMA module has been built and will replace one or both of the lumped mass EMA representations in this model. The resulting analysis will allow the examination of the combined structure while eliminating many of the assumptions used in a quasi-static analysis.

STACK MODEL RESULTS

In these results von-Mises equivalent stress is used for comparison. The design philosophy is to keep all of the stress in the structure below yield. The original shell casing used for SADARM was manufactured from 4340 steel with an approximate yield strength of 195 Kpsi. Subsequently the steel choice for the shell casing was changed to a titanium based maraging-steel – which increased its yield to approximately 250 Kpsi after the final heat treatment. For this analysis the maximum von-Mises stress level was set to red for stresses above 200 Kpsi and pink for levels within 20 Kpsi of the limit. Figure 4 shows the von-Mises stress contours for the 7R and 8S firing conditions at peak pressure in the stack model. The aft submunition is experiencing higher stress levels than the forward submunition. It's not surprising to note that the aft submunition also experiences a higher failure rate. Higher stress levels can also be seen near knock-out holes in the aft lethal mechanism housing and near the IFE ring recess areas.

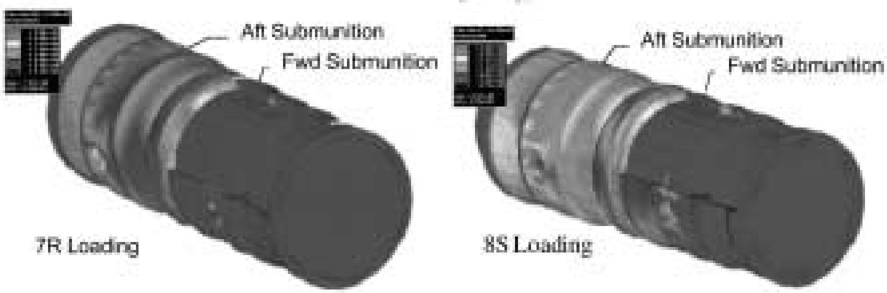

Figure 4. Von-Mises stress contours at peak pressure for stack.

All of the contours provided are at peak pressure. For the stack's structural load-carrying components this is a limiting condition. Although the spin rate presents no real concerns about the lethal mechanism housing and shell casing, the spin load is of concern for the electronic chips and other hardware found in the EMA housing. Additionally, at shot exit the base pressure drops off very rapidly inducing stress waves in the structure.

EMA MODEL

Two configurations of the EMA were under consideration for insertion into the SADARM submunition – potted and unpotted. The circuit card assembly (CCA) is identical in both configurations. In the unpotted configuration, an insert is placed into the EMA along with the CCA. The insert could be metal, plastic or a combination of the two. The insert provides the structural support necessary to ensure the EMA's survival of the gun launch. The insert allows the boards and components to be removed and replaced at any time (a characteristic desired by the Project Manager for production reasons), unlike the potting. The unpotted configuration is the focus of this paper.

Both setback (axial velocity) and spin (rotational velocity) loads were applied to the EMA model. Since these models were feeding into a design effort, computational time was critical. As a result, the initial EMA model of 400,000 brick elements was subdivided into two smaller models. Figure 5 indicates the division between the board 1–2 and the board 3 submodels. Only the components and their surrounding support structure were included in the two models.

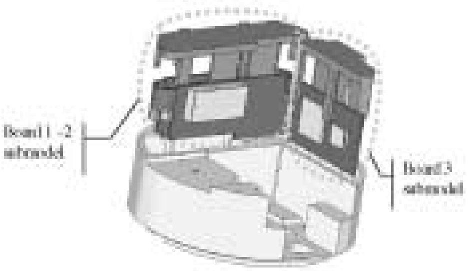

Figure 5. EMA submodels.

The setback and spin loads were applied directly to the support structure material where it would have contacted the EMA housing. The simplification assumes that the EMA housing behaves rigidly and passes all loads through to the boards and their support structures. This is a reasonable assumption since the EMA housing is contained within a fairly rigid steel structure. The run time is significantly reduced, as the housing is no longer modeled.

The primary goal of the EMA model was to determine the loads to which individual components were subjected and their resultant deflections. This deflection is critical since experience demonstrates that it is the main cause of component failure. The model was quite detailed because of this. 1D springs were used to represent the component leads that attached the individual components to the boards. The aggregate stiffness of the leads on an actual component was determined experimentally. The experimentally-derived lead stiffness was then applied to the 1D springs for the simulation.

POST PROCESSING

The EMA analysis examined both dynamic and transient behavior. As a result, the EMA structure spins and translates continuously throughout the model run. In addition, the EMA spin rate is not constant. The EMA is stationary at the beginning and accelerates both axially and angularly, finally achieving a spin in excess of 270 RPS at muzzle exit.

During post processing, the translation is easily removed by forcing the graphical display to auto-center on a single node during the display. A spinning body, however, presents additional complications. Centering a single node on the display is not effective, as

Home
Table of Contents

the body merely appears to rotate about that point. If the rotation is constant, then the coordinate system can be automatically rotated a fixed angle each time step, causing the model to appear stationary. However, in the case of the EMA the rotation is not constant. The principal author developed a specialized algorithm that enables a rotating body to appear stationary during video animation. The net result is that the coordinate system rotates and the spinning body remains stationary, allowing the analyst to concentrate on interpreting the stress contours on the component of interest and not be distracting by spinning bodies.

A basic task in a stress analysis is to plot the peak stress in a component over time. In Griz, there is no provision for finding the maximum stress in a component at each point in time. The principal author developed a script that combines and automates these two processes, greatly simplifying the effort required to output a plot like that shown in Figure 6.

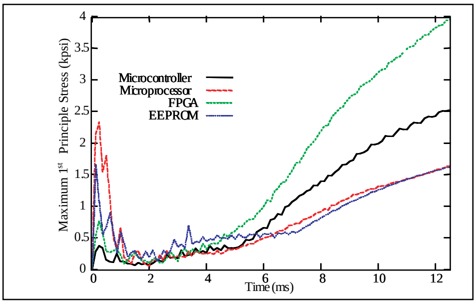

Figure 6. Peak component stress over time.

One specific area of concern for the EMA design was whether board flexure was likely to cause component fracture. The principal author developed another customized routine to extract this information from the model utilizing Griz in conjunction with Matlab. At initialization, the board is undeformed. As the EMA rotates about it’s center of gravity, the board rotates and deforms. The routine implements an algorithm that extracts the out-of-plane flexure of the board in the direction of the arrows, ignoring the rigid motion of the boards.

The algorithm can be used to generate plots like Figure 7. This series of plots graphically illustrates the shape of the board deformation under each component on the left. On the right is a plot showing the magnitude of that deformation. The combination of the two plots allows the analyst to determine if the deformation shape is likely to crack the component and quantify the amount of deformation.

CONCLUSIONS

Finite Element methods (FEM) have proven their worth in analyzing complex structures under varying load conditions. Using FEM, a completed structure can be examined in detail to expose weaknesses in the design. For a number of years FEM have been employed to examine the launch of projectiles from gun system with success. Typical analysis examined the peak pressure loading condition in-bore using a quasi-static balance of pressure acceleration loading. The results proved useful in designing a projectile structure capable of surviving gun launch. The initial design of the SADARM projectile employed these basic techniques. However, when failures that were not predicted in the analysis began to turn up, a closer look was required.

Transient analysis offered the opportunity to look at the spectrum of loading conditions the projectile was being exposed to. The importance of a transient analysis was also examined using simple models. Based on results from these simple models at shot exit and the closeness of the peak stresses to yield, an argument for the use of DYNA3D seems evident. It was demonstrated that the DYNA3D code offered a number of advantages for the analysis of gun-launched projectiles. One of the most important is the sliding interfaces available in the DYNA3D code that allow gaps while preventing inner penetration. These interfaces allow the projectile and gun system to be modeled together and can account for the entire launch cycle. The resulting analysis contains many of the loads not considered in a static FEM approximation – including balloting and transient stress waves during launch and shot exit.

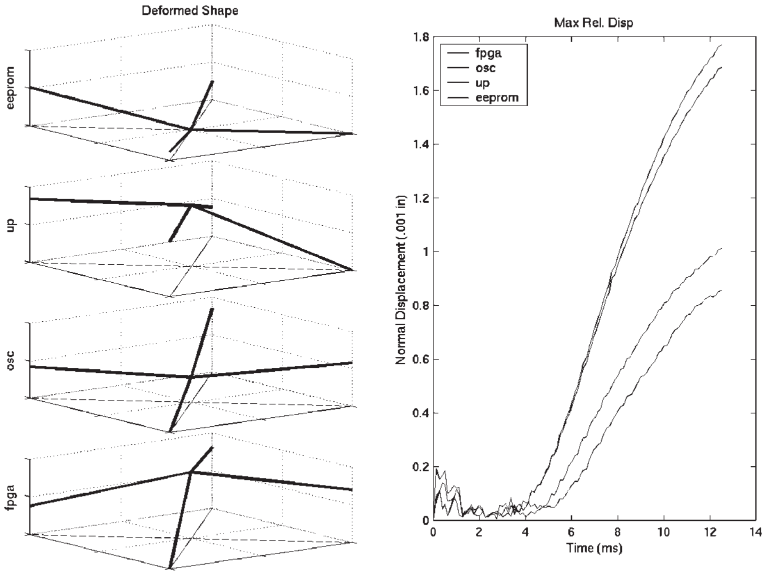

Figure 7. Board deformation plot.

The analysis presented here represents an improvement over conventional FEM static analysis techniques previously used to analyze gun-launched projectiles. This analysis eliminates many of the troubling boundary conditions and assumptions required by a quasi-static analysis. The resulting model is the most accurate and reliable to date. The resulting predictions can be used with a high degree of confidence to improve and optimize current and future smart munitions like SADARM.

A single numerical simulation of an artillery launch of the EMA submunition often takes several days of CPU time and generates a dataset that exceeds several gigabytes. The authors have demonstrated a significant reduction in computational time by a judicious subdivision of the model with appropriate assumptions. Although this subdivision results in a less realistic overall model, it is still quite useful as a design tool. The segmented model allows the analyst to compare a variety of design elements to one another and evaluate their effectiveness.

Once the analysis is complete, the dataset size places a variety of demands on the post-processing tools. In order to effectively analyze the EMA model, algorithms to hold the

Home
Table of Contents

spinning EMA stationary during animations as well as an algorithm capable of extracting the flexure of a spinning circuit board were developed. The reduction in model solution time in conjunction with the additional post-processing tools enabled ARL to provide a finite element analysis of the EMA as part of the SADARM design cycle.

BIBLIOGRAPHY

Wilkerson, S., Hopkins, D., Gazonas, G. and Berman, M. “Developing a Transient Finite Element Model to Simulate the Launch Environment of the 155 mm SADARM Projectile”, ARL-TR-2343, September, 2000.