

## Earth Penetration Simulation using Coupled Eulerian-Lagrangian Analysis

### Summary

In earth penetration events the projectile generally strikes the target at an oblique angle. As a result, the projectile is subjected to a multi-axial force and acceleration history through impact. The effectiveness of an earth penetration system is enhanced by the ability to withstand severe lateral loading. Consequently, it is important to understand how such loads develop during an impact event.

In this Technology Brief, Abaqus/Explicit is used to simulate the impact of a high-strength steel penetrator into a concrete target. The penetrator/target interaction is analyzed using the coupled Eulerian-Lagrangian methodology. Specifically, the penetrator is modeled in a traditional Lagrangian framework while the concrete target is modeled in an Eulerian framework. It will be shown that Abaqus/Explicit results are in good agreement with published experimental data.

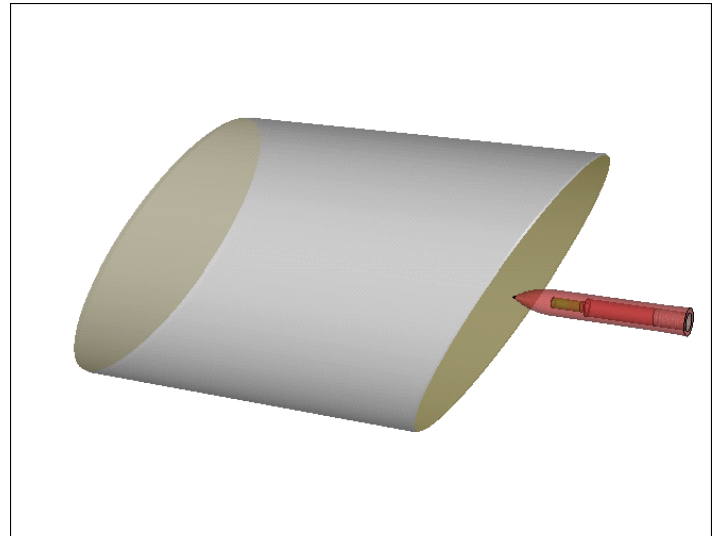
### Background

The in-service accelerations that an earth-penetrating projectile experience can depend on the obliqueness of impact. As the impact condition deviates from the limiting case of being perfectly normal to a flat surface, the accelerations lateral to the projectile's body axis increase.

It is well known that such 'non-normal' impact conditions produce large lateral forces and accelerations that can threaten the survivability of the penetrator case and internal components [1]. Clearly, a predictive capability that allows for simulation of an oblique impact penetration event provides the means to better design earth penetrating systems.

A high-speed impact event very often involves extreme states of deformation and/or material failure. A traditional Lagrangian finite element approach, in which elements deform as the associated material deforms, is limited in the extent of deformation an element can sustain. An Eulerian formulation differs however, in that material flows through a mesh of nodes that remain fixed in space; the Eulerian framework is thus ideally suited for analyses that involve extreme levels of deformation, up to and including material failure and rupture. Abaqus/Explicit allows full coupling of Lagrangian elements and Eulerian material domains in an analysis through simple general contact definitions.

This Technology Brief considers the coupled Eulerian-Lagrangian (CEL) analysis of high-strength steel penetra-



*Click to animate*

### Key Abaqus Features and Benefits

- Coupled Eulerian-Lagrangian capability allows the general contact interaction of Eulerian and Lagrangian regions of a model, simplifying the simulation of the extreme deformation associated with impact events
- Concrete damaged plasticity model simulates damage due to complex stress states
- Accelerometer connector elements measure accelerations in a body-fixed coordinate system
- Anti-aliasing run-time filters simplify the removal of noise from results

tor impacting concrete. As outlined in [1], a series of experiments was undertaken in which concrete test targets were subjected to penetrator impacts at different angles of obliquity. The recorded acceleration results will be compared to those of an Abaqus/Explicit analysis, in which the penetrator is modeled with Lagrangian elements and the concrete is modeled as an Eulerian material.

### Analysis Approach

The penetration test under consideration is illustrated in Figure 1. The concrete target is of a cylindrical shape and is cast in a thin-walled steel pipe. It is cut to an angle

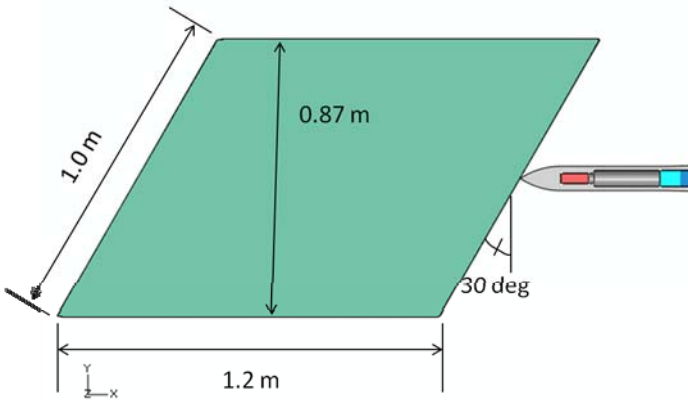


Figure 1: Cross-sectional schematic view of test configuration

of 30 degrees with the cylinder axis.

Figure 2 shows the cross section of the projectile used in the test. Its case is made of 4130 steel. During the test, a smooth-bore powder gun launched the projectile with a striking velocity of 334 m/s. The direction of the impact velocity was aligned with the axis of the cylinder such that the angles of pitch and yaw (i.e. the amount of misalignment between the velocity and projectile axes) were both less than one degree and could be safely neglected.

The projectile's acceleration was measured with a forward mounted accelerometer that collected data at 10 $\mu$ s intervals. The data was filtered with a two-pass, four-pole Butterworth filter with a cutoff frequency of 2 kHz.

#### Finite Element Model

The full three-dimensional projectile is modeled with approximately 13,000 Lagrangian hexahedral reduced-integration elements. The target is modeled with an Eulerian mesh, including both the concrete and the thin-walled steel pipe. The Eulerian mesh is extended beyond the initial volume of concrete so that fragments created during impact are accounted for. A cross section of the assembled model is shown in Figure 3.

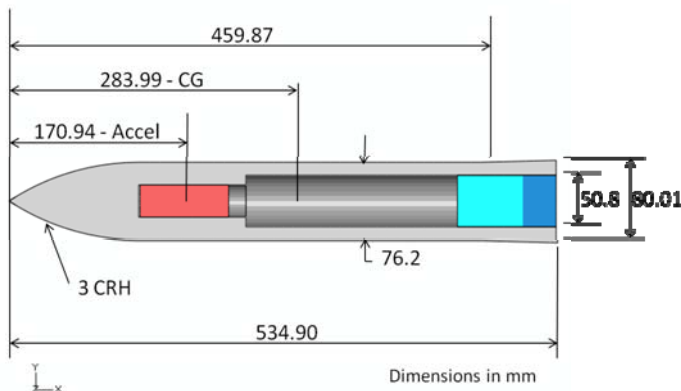


Figure 2: Cross-sectional view of projectile

The objective is to create a mesh that is adequately refined near the contact zone while keeping the total number of elements to a minimum. To achieve this goal, the mesh resolution is biased in such a way that the mesh becomes coarser away from the contact zone. A mesh refinement study was conducted, and a uniform grid spacing of 5 mm in the immediate area of impact was found satisfactory. This 5 mm grid extends out uniformly in both in-plane directions for about 5 projectile diameters, beyond which the grid is graded out to the boundaries. In the direction of penetration, a uniform mesh with a 5 mm resolution is used. The final Eulerian mesh consists of approximately 1 million elements. The duration of the dynamic analysis is 7 milliseconds.

To measure acceleration, an accelerometer-type connector element is placed at the center of the acceleration recording device (shown in Figure 2). This type of connector provides a convenient way to measure acceleration in a coordinate system fixed to a moving body.

#### Material Models

The concrete damaged plasticity model is used to capture the constitutive behavior of the concrete. With the inclusion of multi-hardening plasticity and isotropic damaged elasticity, the model is ideal for applications where concrete is subject to monotonic, cyclic or dynamic loading.

It assumes that the two main failure mechanisms are tensile cracking and compressive crushing. For this analysis, the associated data were derived from hydrostatic and shear failure envelopes given in [1]. While not included in this analysis, the effect of strain-rate dependence can be added for an even higher level of fidelity.

The penetrator case, steel pipe, and penetrator internal components are modeled with linear elastic materials. This is based on the assumption that the amount of abrasion and erosion of the penetrator nose is small and the

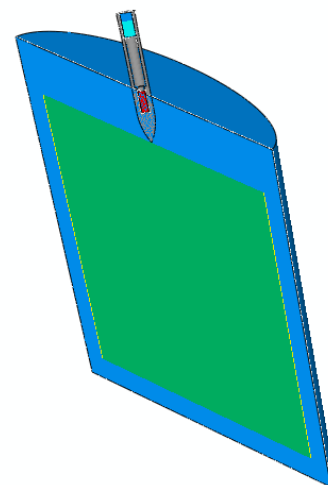


Figure 3: Cross section of assembled coupled Eulerian-Lagrangian model

penetrator wall does not yield. The material properties for the internal components were chosen such that the correct penetrator weight and moments of inertia were achieved.

### Contact Interactions and Loadings

The interaction between the Eulerian target and Lagrangian penetrator is handled through the general contact capability in Abaqus/Explicit, which automatically detects and resolves contact interactions.

### Results

The primary quantities of interest from the test are the axial and lateral projectile accelerations, the depth of penetration, and the rest (turning) angle.

#### Acceleration histories

The time history plot of both the axial (blue) and lateral (green) accelerations is shown in Figure 4. Both the test (dashed) and the simulation (solid) results are shown. Good correlation is observed between the test and simulation results, with peak acceleration difference less than 5%.

Simulation of high-speed impact events with explicit finite element schemes is known to produce noisy results. Anti-aliasing filters are provided in Abaqus (during run time in the solver phase or in the post-processing phase) to conveniently filter the noise. The acceleration data shown in Figures 4 and 6 have been filtered using a sine-Butterworth filter with a cutoff frequency of 2 kHz.

To be consistent with the experimental setup, the axial acceleration direction is aligned with the negative global X

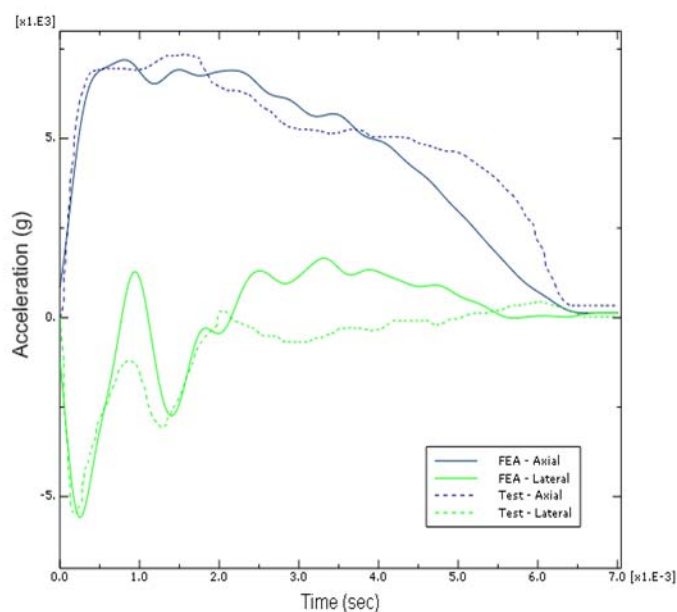


Figure 4: Comparison of Abaqus/Explicit and measured acceleration histories. Note that deceleration is positive.

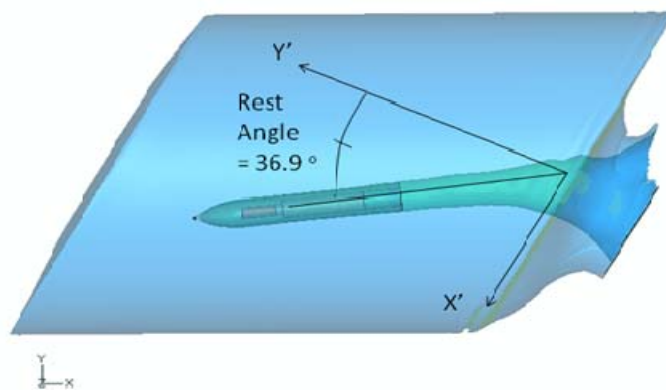


Figure 5: Depth of penetration and rest (turning) angle

direction (Figure 2). This makes the axial deceleration of the projectile positive. The lateral acceleration direction is aligned with the negative global Y direction.

The axial acceleration profile in Figure 4 shows that from initial impact until the penetrator nose is buried, the deceleration increases. After that, the deceleration decreases in the general trend of the experimental data.

Initiated by asymmetric loads (due to the oblique impact) during nose burial, the penetrator pitches upward and eventually the tail impacts the side of the hole created by the burrowing penetrator. The bending of the beam-like penetrator body due to the large inertial loading immediately after the initial impact exacerbates this “slap-down” process. The effect from this significant short-duration lateral loading is evident in the lateral acceleration profile in Figure 4. As the event progresses, secondary lateral loads continue to produce peaks in the profile as the penetrator body contacts the sides of the hole.

#### Depth of Penetration

Results for the final depth of penetration and the rest (turning) angle are shown in Figure 5 and Table 1. As with the acceleration results, there is good correlation between the test and the simulation. Figure 6 contains an animation that provides a synchronized view of the penetration deformation and the acceleration time history. Note that the peak von Mises stress shown is well below the yield strength of 1345 MPa for the 4130 steel penetrator case, except in the region near the nose where localized erosion is expected and was actually observed during the test.

	Nose Tip X' (m)	Nose Tip Y' (m)	Path Length (m)	X'-Y' Rest Angle (degrees)
Test	0.62	0.69	0.93	43.0
FEA	0.60	0.80	1.00	36.9

Table 1: Path length and rest angle results and measurements

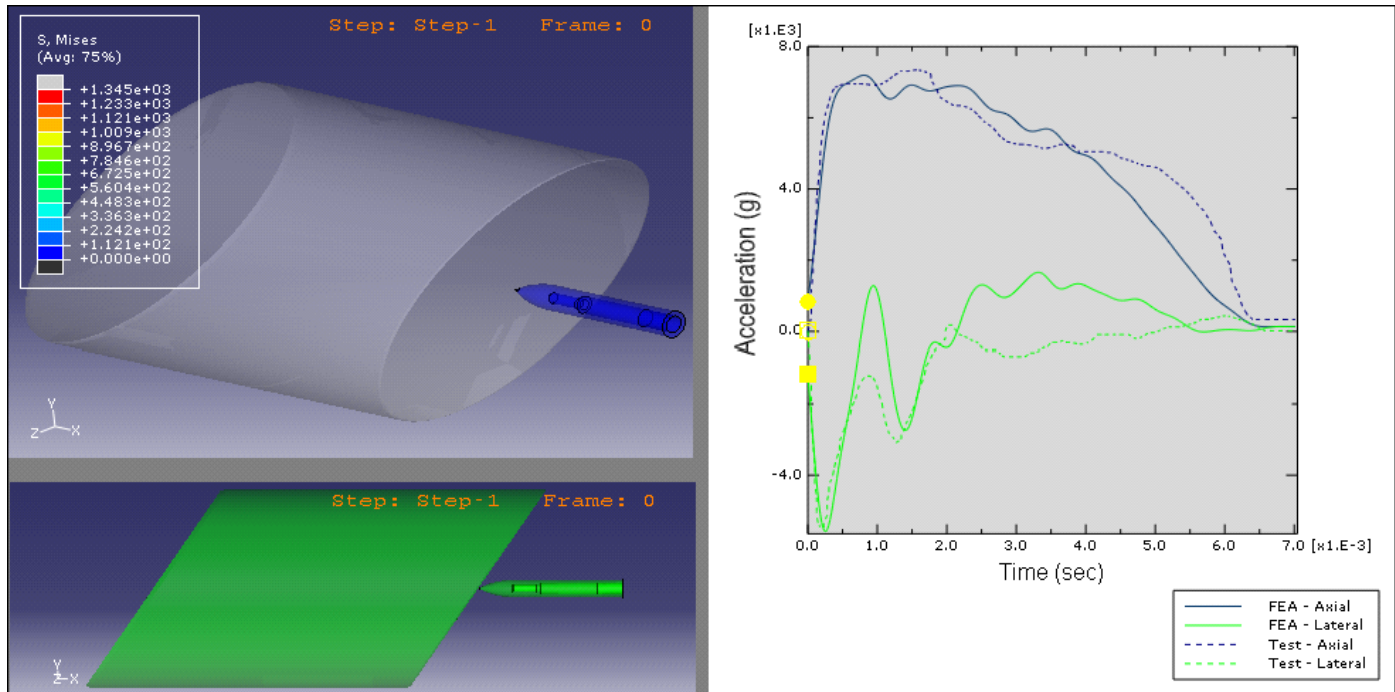


Figure 6: Synchronized animation of deformation, acceleration time histories, and von Mises stress in penetrator case.  
Click to animate

## Summary

A coupled Eulerian-Lagrangian finite element based approach to the simulation of earth penetration events is demonstrated using Abaqus/Explicit. The results from this

approach correlate very well with the chosen experimental data and demonstrate the utility of the coupled approach to modeling high-speed impact events that involve material damage and extreme levels of deformation.

## References

1. Stokes, E., Yarrington, P. and Glenn, L., "An Earth Penetration Modeling Assessment: Prediction of Lateral Loading for Oblique Impact Conditions", UCRL-TR-213206, Lawrence Livermore National Laboratory, Livermore, California, 2005. <https://e-reports-ext.llnl.gov/pdf/321587.pdf>

## Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 references:

- 'Eulerian analysis,' Section 14.1.1 of the Abaqus Analysis User's Manual

## About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit [www.simulia.com](http://www.simulia.com)

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include Abaqus, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright Dassault Systèmes, 2009

