Tech Science Press

ICCES, 2024, vol.31, no.3
DOI: 10.32604/icces.2024.011296

PROCEEDINGS

Development of an Abaqus Plug-in for Designing Hybrid Composite Laminates Against Projectile Impact

Fengbo Han[1,*], Kapil Krishnan[1], Jide Oyebanji[1], Naresh Kakur[1], Rafael Savioli[1], Alia Ruzanna Aziz[1], Henrique Ramos[1], Nikolaos Nikos[1], Rafael Santiago[1] and Zhongwei Guan[1]

[1]Advanced Materials Research Center, Technology Innovation Institute, Abu Dhabi, 9639, United Arab Emirates
[*]Corresponding Author: Fengbo Han. Email: Fengbo.Han@tii.ae

ABSTRACT

This study introduces an innovative plug-in developed within the ABAQUS and Fox GUI environments, which is designed to streamline the design and simulation of hybrid composite laminates for ballistic impact resistance. The plug-in provides an advanced, user-friendly interface for composite laminate design, projectile selection, and ballistic impact simulation parameter configurations. It includes accurately reconstructed models of three projectile types: the tungsten-carbide core projectile M993, the hardened steel core projectile M61, and the lead-core projectile M80, based on scanned data. A distinctive feature of the plug-in is its capacity to facilitate the design of hybrid composite laminates by enabling users to specify the material and thickness for each layer. It incorporates material parameters for a variety of components, including high-strength steels (Mars 600 and Ramor 550), ceramics (B_4C, SiC, AlN, Al_2O_3), and ultra-high molecular weight polyethylene (UHMWPE) grades HB26 and HB210. Furthermore, the plug-in boasts a robust mesh generation capability, offering options for transition mesh or edge-biased mesh strategies to ensure high-quality simulations. Notably, this plug-in extends its usage beyond design and simulation by enabling the creation of extensive database for machine learning studies and structural optimization. This feature makes it a very useful tool for conducting in-depth analyses and developing optimized composite structures tailored to specific ballistic impact scenarios. Case studies, including designs for a steel-steel structure and a ceramic-UHMWPE structure created using the plug-in, underscore its practical applications. Ballistic impact simulations on these structures, compared with experimental data, demonstrate the plug-in's accuracy and effectiveness in predicting the performance of hybrid composite laminates under impact. In summary, the developed plug-in not only simplifies the design and simulation of composite laminates for ballistic protection but also provides a powerful platform for generating data for machine learning and optimization purposes.

KEYWORDS

Hybrid composite laminates; ballistic impact; finite element; Abaqus plug-in

Funding Statement: The authors received no specific funding for this study.

Conflicts of Interest: The authors declare that they have no conflicts of interest to report regarding the present study.