Analysis concrete slab subjected to missile impact using Concrete Damage Plasticity constitutive model in Abaqus/Explicit

Ehsan pedram-Mojtaba Hosseini

Department of Mechanical engineering Branch, Islamic Azad University, Takestan, Iran

Ehsan.pedram7@yahoo.com
Mo-hosseini78@yahoo.com

Abstract:

This paper describes numerical missile impact analysis on a reinforced concrete slab using the Finite Element (FE) solver ABAQUS/Explicit. The FE model of the impacted reinforced concrete slab resembles a structure used in the missile impact tests by Hanchak et al. FE analyses with a hard (rigid) and a soft (deformable) missile and sensitivity studies related to the initial missile velocity are performed. Traditional Lagrangian formulations for both the missiles and reinforced concrete slabs are used. Concrete Damaged Plasticity constitutive Model for concrete in ABAQUS/Explicit, is used and its suitability and limitations for missile impact analyses are explored. The Concrete Damaged Plasticity Model of ABAQUS/Explicit leads to reasonable and sound results in terms of strains/stresses of the reinforced concrete slab, overall energy balances and overall deformation of the concrete slab.

Keywords: Impact loads-Concrete-Abaqus/explicit-Concrete Damage Plasticity-Missile

1- Introduction

The issue of missile impacts on concrete containment buildings (CCBs) of nuclear power plants (NPPs) was subject to intensive research for the first time in the 1970s and early 1980s.During that period a number of missile impact tests, even on a large scale have been carried out, most notably the Meppen Tests in Germany and the Tests at Sandia National Laboratory in the USA. In both tests soft and hard missiles were impacted on large reinforced concrete slabs resembling the CCBs of NPPs build at that time. In parallel quite a number of computational analyses have been performed to predict the results of these tests. For these analyses either empirical formulas or