عنوان مقاله:

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

نویسندگان:

Ehsan pedram - *Department of Mechanical engineering Branch, Islamic Azad University, Takestan, Iran*

Mojtaba Hosseini

خلاصه مقاله:

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 andreinforced 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

کلمات کلیدی:

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