Abstract :

The mechanics of brittle damage in porcelain of an endodontically treated maxilla incisor tooth was simulated using finite element method (FEM). For this purpose a very complex composite structure of endodontically treated tooth is simulated under transverse loading. Three dimensional (3D) model of human maxilla incisor tooth root was developed based on Computed Tomography (CT) scan images. Crown, core cement, resin core, dental post, post cement and dentin were created using SolidWorks software, and then the model was imported into ABAQUS-6.9EF software for nonlinear behavior analysis. This study utilizes finite element method to simulate onset and propagation of crack in ceramic layer (porcelain) by the cause of both tension and compression loading related to complexity of the geometry of tooth implant. The simulation has been done using brittle damaged model available in ABAQUS/Explicit in quasi-static load condition. The load-displacement response of whole structure is measured from the top of porcelain by controlling displacement on a rigid rod. Crack initiated at the top of porcelain bellow the location of the rod caused by tension damage at equivalent load of 590 N. Damage in porcelain accounts for up to 63% reduction of whole structure stiffness from the undamaged state. The failure process in porcelain layer can be described by an exponential rate of fracture energy dissipation. This study demonstrated that the proposed finite element model and analysis procedure can be use to predict the nonlinear behavior of tooth implant.