

1. Record Nr.	UNINA9910366580703321
Autore	Boulbes Raphael Jean
Titolo	Troubleshooting Finite-Element Modeling with Abaqus : With Application in Structural Engineering Analysis // by Raphael Jean Boulbes
Pubbl/distr/stampa	Cham : , : Springer International Publishing : , : Imprint : Springer, , 2020
ISBN	3-030-26740-7
Edizione	[1st ed. 2020.]
Descrizione fisica	1 online resource (XXVII, 439 p. 173 illus., 127 illus. in color.)
Disciplina	620.11 620.00151825
Soggetti	Materials science Mechanics Mechanics, Applied Engineering design Structural materials Computer-aided engineering Characterization and Evaluation of Materials Solid Mechanics Engineering Design Structural Materials Computer-Aided Engineering (CAD, CAE) and Design
Lingua di pubblicazione	Inglese
Formato	Materiale a stampa
Livello bibliografico	Monografia
Nota di contenuto	Introduction -- Analysis convergence guidelines -- Method to debug a model -- General prerequisites -- Materials -- Mesher and meshing -- Contact -- Troubleshooting in job diagnostic -- Numerical acceptance criteria -- Need some help? -- Hardware or software issues -- Index.
Sommario/riassunto	This book gives Abaqus users who make use of finite-element models in academic or practitioner-based research the in-depth program knowledge that allows them to debug a structural analysis model. The book provides many methods and guidelines for different analysis types and modes, that will help readers to solve problems that can

arise with Abaqus if a structural model fails to converge to a solution. The use of Abaqus affords a general checklist approach to debugging analysis models, which can also be applied to structural analysis. The author uses step-by-step methods and detailed explanations of special features in order to identify the solutions to a variety of problems with finite-element models. The book promotes:

- a diagnostic mode of thinking concerning error messages;
- better material definition and the writing of user material subroutines;
- work with the Abaqus mesher and best practice in doing so;
- the writing of user element subroutines and contact features with convergence issues; and
- consideration of hardware and software issues and a Windows HPC cluster solution.

The methods and information provided facilitate job diagnostics and help to obtain converged solutions for finite-element models regarding structural component assemblies in static or dynamic analysis. The troubleshooting advice ensures that these solutions are both high-quality and cost-effective according to practical experience. The book offers an in-depth guide for students learning about Abaqus, as each problem and solution are complemented by examples and straightforward explanations. It is also useful for academics and structural engineers wishing to debug Abaqus models on the basis of error and warning messages that arise during finite-element modelling processing.

---