Bahman Meyghani
Mokhtar Awang

Welding Simulations Using ABAQUS

A Practical Guide for Engineers

Springer

Welding Simulations Using ABAQUS

Bahman Meyghani · Mokhtar Awang

Welding Simulations Using ABAQUS

A Practical Guide for Engineers

Springer

Bahman Meyghani
BKL B.V.
Collse Heide 1
Nuenen, The Netherlands

Department of Mechanical Engineering
Faculty of Engineering Technology
and Built Environment, UCSI University
Kuala Lumpur, Malaysia

Mokhtar Awang
Mechanical Engineering Department
Universiti Teknologi Petronas
Seri Iskandar, Perak, Malaysia

ISBN 978-981-19-1319-8 ISBN 978-981-19-1320-4 (eBook)
https://doi.org/10.1007/978-981-19-1320-4

This Springer imprint is published by the registered company Springer Nature Singapore Pte Ltd.
The registered company address is: 152 Beach Road, #21-01/04 Gateway East, Singapore 189721, Singapore

To our beloved families and friends.
Thank you for all years of love,
encouragement and unfailing support.

Preface

Increasing human needs leads to the creation of complicated scientific problems. Solving a lot of mechanical and industrial problems requires the use of advanced joining methods to join different parts, components, similar and dissimilar metals or some nonmetal materials together. Understanding the behavior of these advanced joining methods by using classical computational methods such as elasticity theory of stress distribution and classical temperature distribution governing equations is challenging. To overcome the difficulties of these challenges, finite element methods are proposed as an appropriate method. In this regard, ABAQUS is well known as a user-friendly commercial finite element software for modeling different processes in mechanical, civil, aerospace and some other engineering fields. As an advanced welding method, Friction Stir Welding (FSW) has a complicated thermomechanical nature, and investigating the process behavior has attracted enormous research interests during the past decades. Numerical simulation is one of the most important methods for digging down deep into the process in detail. This tutorial book contains unified and detailed tutorials for engineers and students who are interested in simulating the FSW process in ABAQUS finite element software.

Nuenen, The Netherlands/Kuala Lumpur, Malaysia
Seri Iskandar, Malaysia

Bahman Meyghani
Mokhtar Awang

Acknowledgements The authors of this book would like to acknowledge Prof. Wallace Kaufman for his endless support.

About ABAQUS Modules

In this section, it is important to explain ABAQUS software modules. The part module is normally used for creating the geometries or importing complicated geometries from other cad software. The property module is used to assign the material, create the section and assign the section to a part. Assembly module is used to instance the part meaning. Step module is used for creating the simulation steps. Interaction module is employed to create the interactions and contact behavior between different parts. Load module is used to define the loads, boundary conditions and predefined parameters such as initial temperature. Finite element meshes are defined in the mesh module. Job module is used to create the input file and define the solver. Visualization module is used to visualize the simulation results. Sketch module is almost the same as part module; however, it can be only employed for sketching.

Contents

List of Figures