

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

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>

© The Editor(s) (if applicable) and The Author(s), under exclusive license to Springer Nature Singapore Pte Ltd. 2022

This work is subject to copyright. All rights are solely and exclusively licensed by the Publisher, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilms or in any other physical way, and transmission or information storage and retrieval, electronic adaptation, computer software, or by similar or dissimilar methodology now known or hereafter developed.

The use of general descriptive names, registered names, trademarks, service marks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

The publisher, the authors and the editors are safe to assume that the advice and information in this book are believed to be true and accurate at the date of publication. Neither the publisher nor the authors or the editors give a warranty, expressed or implied, with respect to the material contained herein or for any errors or omissions that may have been made. The publisher remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

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

1	Introduction to Friction Stir Welding (FSW)	1
	References	2
2	Thermomechanical Analysis	5
	References	6
3	Finite Element Modeling of FSW	7
3.1	Different Modelling Methods	8
3.1.1	CSM Method	8
3.1.2	CFD Method	9
3.1.3	Comparison of Governing Equations for CSM and CFD Methods	10
3.1.4	Computational Approaches	10
	References	11
4	Arbitrary Lagrangian–Eulerian (ALE) Method	13
4.1	Geometry and Preprocessor	13
4.2	Material Property	17
4.3	Assembly of the Model	24
4.4	Step Creation	27
4.5	Interactions	31
4.6	Loads and the Boundary Conditions	43
4.7	Meshing	46
	References	60
5	Post Processor and Visualization of the Results	63
5.1	Post Processor	63
6	Thermomechanical Simulation of FSW Using User Defined Subroutine Modeling Technique	69
6.1	Preparation of the Model	69
6.2	Material Property	69
6.3	Assembly of the Model	70

6.4	Simulation Step Creation	75
6.5	Interaction Definitions	79
6.6	Mesh Module	82
6.7	Loads and Boundary Conditions	87
6.8	Postprocessor (Thermal Model)	98
6.9	Mechanical Step	101
6.10	Loads and Boundary Conditions	105
6.11	Mesh Module	107
6.12	Returning to the Step Module in Order to Request the Results	109
6.13	Postprocessor	109
	References	122
	Conclusions	123

List of Figures

Fig. 1.1	Schematic view of the FSW process [18]	2
Fig. 4.1	Comparison between the main domain and the ALE	14
Fig. 4.2	Creating a new part	14
Fig. 4.3	Creating a rectangle	15
Fig. 4.4	Defining rectangle points	15
Fig. 4.5	Selecting material thickness	15
Fig. 4.6	Creating part	16
Fig. 4.7	Tool dimensions	17
Fig. 4.8	Revolving the tool	17
Fig. 4.9	Selecting the tool as a rigid reference	18
Fig. 4.10	Creating shell	18
Fig. 4.11	Selecting the tool	18
Fig. 4.12	Approving the selected body	18
Fig. 4.13	Creating the material	19
Fig. 4.14	Defining thermal conductivity	19
Fig. 4.15	Applying the density	20
Fig. 4.16	Defining the elastic behaviour	20
Fig. 4.17	Defining the expansion	21
Fig. 4.18	Selecting the inelastic heat fraction	21
Fig. 4.19	Defining the latent heat	21
Fig. 4.20	Defining the plasticity	21
Fig. 4.21	Defining the rate dependent	22
Fig. 4.22	Defining the specific heat	23
Fig. 4.23	Creating the solid homogenous section	23
Fig. 4.24	Assigning the section	23
Fig. 4.25	Selecting the region	24
Fig. 4.26	Editing the assignment	24
Fig. 4.27	Creating the assembly	25
Fig. 4.28	Rotating the position	25
Fig. 4.29	Setting the position	26
Fig. 4.30	Corrected position of the workpiece	26

Fig. 4.31	Changing the position of the tool	27
Fig. 4.32	Setting the tool position	27
Fig. 4.33	Correction position for the tool and the workpiece	28
Fig. 4.34	Partitioning the workpiece	28
Fig. 4.35	Selecting the line and creating the rectangle	28
Fig. 4.36	Drawing the rectangle for the partitioning	29
Fig. 4.37	Create partitions	29
Fig. 4.38	Selecting the edge	29
Fig. 4.39	Selecting the edges and extrude it along the direction	30
Fig. 4.40	Selecting the line	30
Fig. 4.41	Creating the partition	30
Fig. 4.42	Partitining the face	31
Fig. 4.43	Selecting the vertical line	31
Fig. 4.44	Selecting the rectangle	31
Fig. 4.45	Rectangle in another position	32
Fig. 4.46	Selecting extrude/sweep edges	32
Fig. 4.47	Selecting the cells	32
Fig. 4.48	Selecting the rectangle	32
Fig. 4.49	Approving the selected region	33
Fig. 4.50	Showing the preview of the partitioned region	33
Fig. 4.51	Creating the partition	33
Fig. 4.52	Partitioned workpiece	34
Fig. 4.53	Creating the step	34
Fig. 4.54	Select the step time	35
Fig. 4.55	Applying the mass scaling	35
Fig. 4.56	Applying the ALE adaptive mesh	36
Fig. 4.57	Selecting ALE region	36
Fig. 4.58	Approving the region	36
Fig. 4.59	Confirming the region	37
Fig. 4.60	Approving the rigid body	37
Fig. 4.61	Selecting the tool	38
Fig. 4.62	Confirming the region	38
Fig. 4.63	Selecting the tool set	38
Fig. 4.64	Confirming the rigid body	39
Fig. 4.65	Applying the friction coefficient	39
Fig. 4.66	Applying the heat generation	40
Fig. 4.67	Contact surface	40
Fig. 4.68	Approving the selected region	41
Fig. 4.69	Selected region in the workpiece	41
Fig. 4.70	Approving the region	41
Fig. 4.71	Confirming the interaction part	42
Fig. 4.72	Creating the boundary condition	43
Fig. 4.73	Selecting the region	43
Fig. 4.74	Approving the region	44
Fig. 4.75	Applying the boundary condition	44

Fig. 4.76	Creating the boundary condition	45
Fig. 4.77	Confirming the boundary condition	45
Fig. 4.78	Selecting the limitations of the boundary conditions	45
Fig. 4.79	Applying the boundary condition	46
Fig. 4.80	Velocity boundary condition	46
Fig. 4.81	Create boundary condition	47
Fig. 4.82	Selecting the region	47
Fig. 4.83	Applying the boundary condition	47
Fig. 4.84	Applying the amplitude	48
Fig. 4.85	Creating the predefined field	49
Fig. 4.86	Selecting the set	49
Fig. 4.87	Selecting the set	50
Fig. 4.88	Applying the value of 25	50
Fig. 4.89	Creating the predefined field	51
Fig. 4.90	Selecting the set	51
Fig. 4.91	Selecting the set	51
Fig. 4.92	Applying the value of 25	52
Fig. 4.93	Selecting the part	52
Fig. 4.94	Applying the part instances	52
Fig. 4.95	Global seeds selection	53
Fig. 4.96	Selecting the part instances	53
Fig. 4.97	Applying the seed edges	53
Fig. 4.98	Selecting the assigned region	53
Fig. 4.99	Selecting the local seeds	54
Fig. 4.100	Assigning the region	54
Fig. 4.101	Selecting the region	54
Fig. 4.102	Applying the part instances	55
Fig. 4.103	Assigning the element type for the workpiece	55
Fig. 4.104	Selecting the region	55
Fig. 4.105	Selecting the element type	56
Fig. 4.106	Applying the region	56
Fig. 4.107	Seed part instances	56
Fig. 4.108	Approving the region	57
Fig. 4.109	Selecting the size of the mesh	57
Fig. 4.110	Selecting the part instances	57
Fig. 4.111	Mesh part instances	57
Fig. 4.112	Approving the region	58
Fig. 4.113	Assigning the element type for the tool	58
Fig. 4.114	Selecting the set	58
Fig. 4.115	Selecting the set between sets	58
Fig. 4.116	Applying the explicit element type	59
Fig. 4.117	Selecting the set	59
Fig. 4.118	Confirming the region	59
Fig. 5.1	Creating the job	64
Fig. 5.2	Editing the job manager	64

Fig. 5.3	Running the job	65
Fig. 5.4	Completed job	65
Fig. 5.5	Stress across the plate	66
Fig. 5.6	Nodal temperature	66
Fig. 5.7	Equivalent plastic strain	67
Fig. 6.1	Creating the new model	70
Fig. 6.2	Creating the part	70
Fig. 6.3	Creating the rectangle	71
Fig. 6.4	Rectangle dimentions	71
Fig. 6.5	Extrusion values	71
Fig. 6.6	Final part	72
Fig. 6.7	Creating the material property	72
Fig. 6.8	Material property definition	73
Fig. 6.9	Creating the solid homogenous section	74
Fig. 6.10	Assigning the section	74
Fig. 6.11	Confirming the region	74
Fig. 6.12	Editing the section assignment	75
Fig. 6.13	Creating the instance part	75
Fig. 6.14	The created part	76
Fig. 6.15	Selection the rotation point	76
Fig. 6.16	The rotation angle	76
Fig. 6.17	Confirming the rotation	76
Fig. 6.18	Positioning	77
Fig. 6.19	Creating the instance	77
Fig. 6.20	Selecting the point	78
Fig. 6.21	Selecting the point for the coordinate system	78
Fig. 6.22	Confirming the position of the instance	78
Fig. 6.23	Creating the step	79
Fig. 6.24	Creating the second step	80
Fig. 6.25	Requesting the filed output	81
Fig. 6.26	Requesting the filed output	81
Fig. 6.27	Requesting the restart	82
Fig. 6.28	Defining the interaction	82
Fig. 6.29	Defining the surface film condition	83
Fig. 6.30	Selecting the surface	83
Fig. 6.31	Editing the interaction	84
Fig. 6.32	Creating the interaction	84
Fig. 6.33	Selecting the surface	85
Fig. 6.34	Editing the interaction	85
Fig. 6.35	Creating the interaction	86
Fig. 6.36	Selecting the surface	86
Fig. 6.37	Editing the interaction	87
Fig. 6.38	Selecting the seed edge	88
Fig. 6.39	Selecting the seed edge	89
Fig. 6.40	Selecting the surface	90

Fig. 6.41	Selecting the heat transfer element	91
Fig. 6.42	Creating the mesh	92
Fig. 6.43	Creating the predefined field	92
Fig. 6.44	Defining the initial temperature	93
Fig. 6.45	Selecting of the workpiece	93
Fig. 6.46	Approving the selected workpiece	94
Fig. 6.47	Selecting the temperature	94
Fig. 6.48	Applying the subroutine	95
Fig. 6.49	Applying the load	96
Fig. 6.50	Deactivation of the step	97
Fig. 6.51	Creating the job	98
Fig. 6.52	Creating the thermal model	99
Fig. 6.53	Defining the subroutine for the simulation	99
Fig. 6.54	Managing the job	100
Fig. 6.55	Job manager part	100
Fig. 6.56	Requesting to continue the job	100
Fig. 6.57	Completed job	101
Fig. 6.58	Heat flux results	101
Fig. 6.59	Element temperature results	102
Fig. 6.60	Nodal temperature results	102
Fig. 6.61	Copying the model	103
Fig. 6.62	Creating the mechanical model	103
Fig. 6.63	Checking the model	103
Fig. 6.64	Creating the step	103
Fig. 6.65	Deleting the step	104
Fig. 6.66	Creating the mechanical step	104
Fig. 6.67	Replacing the step	105
Fig. 6.68	Modifying the step	106
Fig. 6.69	Modifying the step	106
Fig. 6.70	Creating the boundary condition	107
Fig. 6.71	Creating the boundary condition	107
Fig. 6.72	Selecting the side of the workpiece	108
Fig. 6.73	Confirming the set	108
Fig. 6.74	Selecting the boundary condition	108
Fig. 6.75	Selecting the workpiece	109
Fig. 6.76	Creating the boundary condition	109
Fig. 6.77	Selecting the boundary condition	110
Fig. 6.78	Selecting the workpiece	110
Fig. 6.79	Confirming the set	110
Fig. 6.80	Selecting the boundary condition	111
Fig. 6.81	Selecting the boundary condition in the workpiece	111
Fig. 6.82	Creating the predefined field	112
Fig. 6.83	Selecting the temperature	112
Fig. 6.84	Selecting the workpiece	113
Fig. 6.85	Confirming the set	113