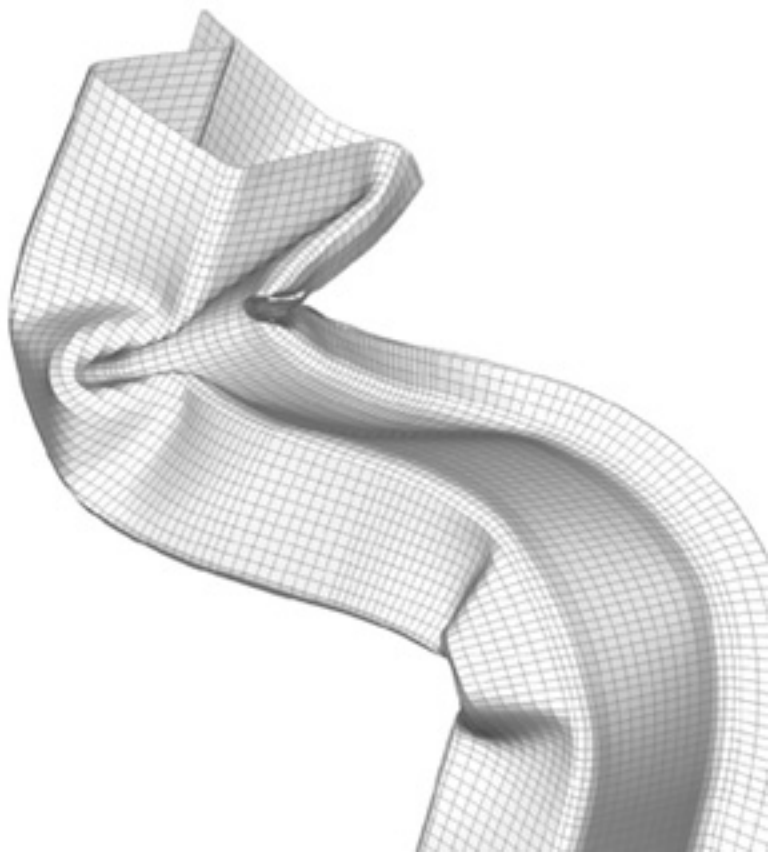


ABAQUS Interface for MSC.ADAMS User's Manual



Trademarks and Legal Notices

CAUTIONARY NOTICE TO USERS:

This manual is intended for qualified users who will exercise sound engineering judgment and expertise in the use of the ABAQUS Software. The ABAQUS Software is inherently complex, and the examples and procedures in this manual are not intended to be exhaustive or to apply to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and results of their analyses.

ABAQUS, Inc. will not be responsible for the accuracy or usefulness of any analysis performed using the ABAQUS Software or the procedures, examples, or explanations in this manual. ABAQUS, Inc. shall not be responsible for the consequences of any errors or omissions that may appear in this manual.

ABAQUS, INC. DISCLAIMS ALL EXPRESS OR IMPLIED REPRESENTATIONS AND WARRANTIES, INCLUDING ANY IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE OF THE CONTENTS OF THIS MANUAL.

IN NO EVENT SHALL ABAQUS, INC. OR ITS THIRD-PARTY PROVIDERS BE LIABLE FOR ANY INDIRECT, INCIDENTAL, PUNITIVE, SPECIAL, OR CONSEQUENTIAL DAMAGES (INCLUDING WITHOUT LIMITATION DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, OR LOSS OF BUSINESS INFORMATION) EVEN IF ABAQUS, INC. HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

The ABAQUS Software described in this manual is available only under license from ABAQUS, Inc. and may be used or reproduced only in accordance with the terms of such license.

This manual and the software described in this manual are subject to change without prior notice.

No part of this manual may be reproduced or distributed in any form without prior written permission of ABAQUS, Inc.

©ABAQUS, Inc. 2004. All rights reserved.

Printed in the United States of America.

U.S. GOVERNMENT USERS: The ABAQUS Software and its documentation are “commercial items,” specifically “commercial computer software” and “commercial computer software documentation,” and consistent with FAR 12.212 and DFARS 227.7202, as applicable, are provided under license to the U.S. Government, with restricted rights.

TRADEMARKS

The trademarks and service marks (“trademarks”) in this manual are the property of ABAQUS, Inc. or third parties. You are not permitted to use these trademarks without the prior written consent of ABAQUS, Inc. or such third parties.

The following are trademarks or registered trademarks of ABAQUS, Inc. or its subsidiaries in the United States and/or other countries: ABAQUS, ABAQUS/Standard, ABAQUS/Explicit, ABAQUS/CAE, ABAQUS/Viewer, ABAQUS/Aqua, ABAQUS/Design, ABAQUS/Foundation, and the ABAQUS Logo.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information, see the Trademark and License Notices in the ABAQUS Version 6.5 Release Notes.

ABAQUS Offices and Representatives

ABAQUS, Inc.
Rising Sun Mills
166 Valley Street
Providence, RI 02909-2499
Tel: +1 401 276 4400
Fax: +1 401 276 4408
E-mail: support@Abaqus.com
<http://www.abaqus.com>

ABAQUS Europe BV
Gaetano Martinolaan 95
P. O. Box 1637
6201 BP Maastricht
The Netherlands
Tel: +31 43 356 6906
Fax: +31 43 356 6908
E-mail: info.europe@abaqus.com

Sales, Support, and Services

UNITED STATES

ABAQUS Central, Inc.
1440 Innovation Place
West Lafayette, IN 47906-1000
Tel: +1 765 497 1373
Fax: +1 765 497 4444
E-mail: support@AbaqusCentral.com

ABAQUS East, LLC
300 Centerville Road, Suite 209W
Warwick, RI 02886-0201
Tel: +1 401 739 3637
Fax: +1 401 739 3302
E-mail: support@AbaqusEast.com

ABAQUS Erie, Inc.
3601 Green Road, Suite 316
Beachwood, OH 44122
Tel: +1 216 378 1070
Fax: +1 216 378 1072
E-mail: support@AbaqusErie.com

ABAQUS Great Lakes, Inc.
14500 Sheldon Road, Suite 160
Plymouth, MI 48170-2408
Tel: +1 734 451 0217
Fax: +1 734 451 0458
E-mail: support@AbaqusGreatLakes.com

ABAQUS South, Inc.
3700 Forums Drive, Suite 101
Flower Mound, TX 75028
Tel: +1 214 513 1600
Fax: +1 214 513 1700
E-mail: support@AbaqusSouth.com

ABAQUS West, Inc.
39221 Paseo Padre Parkway, Suite F
Fremont, CA 94538-1611
Tel: +1 510 794 5891
Fax: +1 510 794 1194
E-mail: support@AbaqusWest.com

ARGENTINA

KB Engineering S. R. L.
Florida 274 - Oficina 35
1005 Buenos Aires
Argentina
Tel: +54 11 4326 9176/7542
Fax: +54 11 4326 2424
E-mail: sanchezsarmiento@arnet.com.ar

AUSTRALIA

Worley Advanced Analysis
Level 17, 300 Flinders Street
Melbourne, Vic 3000
Tel: +61 3 8612 5132
Fax: +61 3 9205 0573
E-mail: abaqus@worley.com.au

AUSTRIA

ABAQUS Austria GmbH
Zinckgasse 20-22/2/13
A-1150 Vienna
Austria
Tel: +43 1 929 16 25-0
Fax: +43 1 929 16 25-20
E-mail: support@abaqus.at

CHINA

ABAQUS China
Room A-2703, Eagle Plaza
No. 26 Xiao Yun Rd.
Beijing, 100016
P. R. China
Tel: +86 01 84580366
Fax: +86 01 84580360
E-mail: abaqus@abaqus.com.cn

FRANCE

ABAQUS France SAS
7 rue Jean Mermoz, Bat. A
78000 Versailles
Tel: +33 01 39 24 15 40
Fax: +33 01 39 24 15 45
E-mail: support@abaqus.fr

GERMANY (Munich)

ABAQUS Deutschland GmbH
Sendlinger-Tor-Platz 8
D-80336 München
Tel: +49 89 5999 1768
Fax: +49 89 5999 1767
E-mail: info@abaqus.de

ITALY

ABAQUS Italia s.r.l.
Via Domodossola, 17
20145 Milano (MI)
Tel: +39 02 39211211
Fax: +39 02 31800064
E-mail: info@abaqus.it

BENELUX

ABAQUS Benelux BV
Huizermaatweg 576
1276 LN Huizen
The Netherlands
Tel: +31 35 52 58 424
Fax: +31 35 52 44 257
E-mail: support@abaqus.nl

CZECH REPUBLIC

Synerma s. r. o.
Huntirov 58
468 22 Skuhrov
Czech Republic
Tel: +420 603 145 769
Fax: +420 603 181 944
E-mail: abaqus@synerma.cz

GERMANY (Aachen)

ABAQUS Deutschland GmbH
Theaterstraße 30-32
D-52062 Aachen
Tel: +49 241 474010
Fax: +49 241 4090963
E-mail: info@abaqus.de

INDIA (Chennai)

ABAQUS Engineering India (Pvt.) Ltd.
3M, Prince Arcade
22-A Cathedral Road
Chennai, 600 086
Tel: +91 44 28114624
Fax: +91 44 28115087
E-mail: abaqus@abaqus.co.in

JAPAN (Tokyo)

ABAQUS, Inc.
3rd Floor, Akasaka Nihon Building
5-24, Akasaka 9-chome, Minato-ku
Tokyo, 107-0052
Tel: +81 3 5474 5817
Fax: +81 3 5474 5818
E-mail: tokyo@abaqus.jp

JAPAN (Osaka)

ABAQUS, Inc.
9th Floor, Higobashi Watanabe Building
6-10, Edobori 1-chome, Nishi-ku
Osaka, 550-0002
Tel: +81 6 4803 5020
Fax: +81 6 4803 5021
E-mail: osaka@abaqus.jp

MALAYSIA

Worley Advanced Analysis
19th Floor, Empire Tower
City Square Centre
182 Jalan Tun Razak
50400 Kuala Lumpur
Tel: +60 3 2161 2266
Fax: +60 3 2161 4266
E-mail: abaqus.my@worley.com.au

POLAND

BudSoft Sp. z o.o.
61-807 Poznań
Sw. Marcin 58/64
Tel: +48 61 8508 466
Fax: +48 61 8508 467
E-mail: budsoft@budsoft.com.pl

SINGAPORE

Worley Advanced Analysis
491B River Valley Road
#09-01 Valley Point
Singapore, 248373
Tel: +65 6735 8444
Fax: +65 6735 7444
E-mail: abaqus.sg@worley.com.au

KOREA

ABAQUS Korea, Inc.
Suite 306, Sambo Building
13-2 Yoido-Dong, Youngdeungpo-ku
Seoul, 150-010
Tel: +82 2 785 6707
Fax: +82 2 785 6709
E-mail: info@abaqus.co.kr

NEW ZEALAND

Matrix Applied Computing Ltd.
P. O. Box 56-316, Auckland
Courier: Unit 2-5, 72 Dominion Road, Mt Eden,
Auckland
Tel: +64 9 623 1223
Fax: +64 9 623 1134
E-mail: abaqus-tech@matrix.co.nz

RUSSIA, BELARUS & UKRAINE

TESIS Ltd.
Office 701-703,
18, Unnatov Str.
127083 Moscow, Russia
Tel: +7 095 212-44-22
Fax: +7 095 212-42-62
E-mail: info@tesis.com.ru

SOUTH AFRICA

Finite Element Analysis Services (Pty) Ltd.
Unit 4, The Waverley
Wycroft Road
Mowbray 7700
Tel: +27 21 448 7608
Fax: +27 21 448 7679
E-mail: feas@feas.co.za

SPAIN

Principia Ingenieros Consultores, S.A.
Velázquez, 94
E-28006 Madrid
Tel: +34 91 209 1482
Fax: +34 91 575 1026
E-mail: abaqus@principia.es

TAIWAN

APIC
11F, No. 71, Sung Chiang Road
Taipei, 10428
Tel: +886 02 25083066
Fax: +886 02 25077185
E-mail: apic@apic.com.tw

TURKEY

A-Ztech Ltd.
Perdemsac Plaza, Teknoloji Evi
Bayar Cad., Gulbahar Sok., No: 17
Kozyatagi
34742 Istanbul
TURKIYE
Tel: +90 216 361 8850
Fax: +90 216 361 8851
E-mail: info@a-ztech.com.tr

SWEDEN

ABAQUS Scandinavia
FEM-Tech AB
Pilgatan 8c
SE-72130 Västerås
Tel: +46 21 12 64 10
Fax: +46 21 18 12 44
E-mail: abaqus@abaqus.se

THAILAND

Worley Advanced Analysis
333 Lao Peng Nguan 1 Building
20th Floor Unit B
Soi Chaypuang
Vibhavadi-Rangsit Road
Ladyao, Jatujak
Bangkok 10900
Thailand
Tel: +66 2 689 3000
Fax: +66 2 618 8109
E-mail: abaqus.th@worley.com.au

UNITED KINGDOM (Cheshire)

ABAQUS UK Ltd.
The Genesis Centre
Science Park South, Birchwood
Warrington, Cheshire WA3 7BH
Tel: +44 1 925 810166
Fax: +44 1 925 810178
E-mail: hotline@abaqus.co.uk

Sales Only

UNITED STATES

ABAQUS East, LLC, Mid-Atlantic Office
114 Zachary Court
Forest Hill, MD 21050
Tel: +1 410 420 8587
Fax: +1 410 420 8908
E-mail: support@AbaqusEast.com

ABAQUS South, Inc., Southeast Office
484 Broadstone Way
Acworth, GA 30101
Tel: +1 770 795 0960
Fax: +1 770 795 7614
E-mail: support@AbaqusSouth.com

ABAQUS West, Inc., Southern CA and AZ Office
1100 Irvine Boulevard #248
Tustin, CA 92780
Tel: +1 714 731 5895
Fax: +1 714 242 7002
E-mail: Info@AbaqusWest.com

ABAQUS West, Inc., Rocky Mountains Office
6910 Cordwood Ct.
Boulder, CO 80301
Tel: +1 303 664 5444
Fax: +1 303 200 9481
E-mail: Info@AbaqusWest.com

FINLAND

ABAQUS Finland Oy
Tekniikantie 12
FIN-02150 Espoo
Tel: +358 9 2517 2973
Fax: +358 9 2517 2200
E-mail: abaqus@abaqus.se

INDIA (Pune)

ABAQUS Engineering Analysis Solutions (Pvt.) Ltd.
C-9, 3rd Floor
Bramha Estate, Kondwa Road
Pune-411040
Tel: +91 20 30913739
E-mail: abaqus@abaqus.co.in

UNITED KINGDOM (Kent)

ABAQUS UK Ltd.
Great Hollenden Business Centre, Unit A
Mill Lane, Underriver
Sevenoaks, Kent TN15 OSQ
Tel: +44 1 732 834930
Fax: +44 1 732 834720
E-mail: hotline@abaqus.co.uk

Preface

This section lists various resources that are available for help with using ABAQUS, including technical engineering and systems support, training seminars, and documentation.

Support

ABAQUS, Inc., offers both technical engineering support and systems support for ABAQUS. Technical engineering and systems support are provided through the nearest local support office. You can contact our offices by telephone, fax, electronic mail, the ABAQUS web-based support system, or regular mail. Information on how to contact each office is listed in the front of each ABAQUS manual. The ABAQUS Online Support System (AOSS) is accessible through the **MY ABAQUS** section of the ABAQUS Home Page (www.abaqus.com). When contacting your local support office, please specify whether you would like technical engineering support (you have encountered problems performing an ABAQUS analysis or creating a model in ABAQUS) or systems support (ABAQUS will not install correctly, licensing does not work correctly, or other hardware-related issues have arisen).

The ABAQUS Online Support System has a knowledge database of ABAQUS Answers. The ABAQUS Answers are solutions to questions that we have had to answer or guidelines on how to use ABAQUS. We welcome any suggestions for improvements to the support program or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to file a complaint about the service or products provided by ABAQUS, refer to the ABAQUS Home Page.

Technical engineering support

ABAQUS technical support engineers can assist in clarifying ABAQUS features and checking errors by giving both general information on using ABAQUS and information on its application to specific analyses. If you have concerns about an analysis, we suggest that you contact us at an early stage, since it is usually easier to solve problems at the beginning of a project rather than trying to correct an analysis at the end.

Please have the following information ready before calling the technical engineering support hotline, and include it in any written contacts:

- Your site identifier, which can be obtained by typing **abaqus whereami** at your system prompt (or by selecting **Help→On Version** from the main menu bar in ABAQUS/CAE or ABAQUS/Viewer).
- The version of ABAQUS that are you using.
 - The version numbers for ABAQUS/Standard and ABAQUS/Explicit are given at the top of the data (**.dat**) file.
 - The version numbers for ABAQUS/CAE and ABAQUS/Viewer can be found by selecting **Help→On Version** from the main menu bar.
 - The version numbers for the ABAQUS Interface for MOLDFLOW and the ABAQUS Interface for MSC.ADAMS are output to the screen.
 - The version number for ABAQUS for CATIA V5 can be found by selecting **Help→About ABAQUS for CATIA V5** from the main menu bar in either of the ABAQUS for CATIA V5 workbenches.

- The type of computer on which you are running ABAQUS.
- The symptoms of any problems, including the exact error messages, if any.
- Workarounds or tests that you have already tried.

When calling for support about a specific problem, any available ABAQUS output files may be helpful in answering questions that the support engineer may ask you.

The support engineer will try to diagnose your problem from the model description and a description of the difficulties you are having. The support engineer may need model sketches, which can be sent via fax, e-mail, or regular mail. Plots of the final results or the results near the point that the analysis terminated may also be needed to understand what may have caused the problem.

If the support engineer cannot diagnose your problem from this information, you may be asked to supply the input data. The data can be attached to a support incident in the ABAQUS Online Support System. It may also be sent by means of e-mail, tape, disk, or ftp. Please check the ABAQUS Home Page (<http://www.abaqus.com>) for the media formats that are currently accepted.

All support incidents are tracked in the ABAQUS Online Support System. This enables you (as well as the support engineer) to monitor the progress of a particular problem and to check that we are resolving support issues efficiently. To use the ABAQUS Online Support System, you need to register with the system. Visit the **MY ABAQUS** section of the ABAQUS Home Page for instructions on how to register. If you are contacting us by means outside the AOSS to discuss an existing support problem and you know the incident number, please mention it so that we can consult the database to see what the latest action has been and, thus, give you more efficient support as well as avoid duplication of effort. In addition, please give the receptionist the support engineer's name if contacting us via telephone or include it at the top of any e-mail correspondence.

Systems support

ABAQUS systems support engineers can help you resolve issues related to the installation and running of ABAQUS, including licensing difficulties, that are not covered by technical engineering support.

You should install ABAQUS by carefully following the instructions in the ABAQUS Installation and Licensing Guide. If you are able to complete the installation, please make sure that the product verification procedure was run successfully at the end of the installation procedure. Successful verification for licensed products would indicate that you can run these products on your computer; unsuccessful verification for licensed products indicates problems with the installation or licensing (or both). If you encounter problems with the installation, licensing, or verification, first review the instructions in the ABAQUS Installation and Licensing Guide to ensure that they have been followed correctly. If this does not resolve the problems, consult the ABAQUS Answers database in the ABAQUS Online Support System for information about known installation problems. If this does not address your situation, please create an incident in the AOSS and describe your problem, including the output from **abaqus info=support**. If you call, mail, e-mail, or fax us about a problem (instead of using the AOSS), please provide the output from **abaqus info=support**. It is important that you provide as much information as possible about your problem: error messages from an aborted analysis, output from the **abaqus info=support** command, etc.

ABAQUS Web server

For users connected to the Internet, many questions can be answered by visiting the ABAQUS Home Page on the World Wide Web at

`http://www.abaqus.com`

The information available on the ABAQUS Home Page includes:

- Link to the AOSS
- ABAQUS systems information and computer requirements
- ABAQUS performance data
- Error status reports
- ABAQUS documentation price list
- Training seminar schedule
- ABAQUS Insights newsletter
- Technology briefs

Anonymous ftp site

For users connected to the Internet, ABAQUS maintains useful documents on an anonymous ftp account on the computer ftp.abaqus.com. Simply ftp to ftp.abaqus.com. Login as user anonymous, and type your e-mail address as your password. Directions will come up automatically upon login.

Writing to technical support

Address of ABAQUS Headquarters:

ABAQUS, Inc.
166 Valley Street
Providence, RI 02909, USA
Attention: Technical Support

Addresses for other offices and representatives are listed in the front of each manual.

Support for academic institutions

Under the terms of the Academic License Agreement we do not provide support to users at academic institutions. Academic users can purchase technical support on an hourly basis. For more information, please see the ABAQUS Home Page or contact your local ABAQUS support office.

Training

All ABAQUS offices offer regularly scheduled public training classes.

The Introduction to ABAQUS seminar covers basic modeling using ABAQUS/CAE and linear and nonlinear applications, such as large deformation, plasticity, contact, and dynamics using ABAQUS/Standard and ABAQUS/Explicit. Workshops provide as much practical experience with ABAQUS as possible.

Advanced seminars cover topics of interest to customers with experience using ABAQUS, such as engine analysis, metal forming, fracture mechanics, and heat transfer.

We also provide training seminars at customer sites. On-site training seminars can be one or more days in duration, depending on customer requirements. The training topics can include a combination of material from our introductory and advanced seminars. Workshops allow customers to exercise ABAQUS on their own computers.

For a schedule of seminars, see the ABAQUS Home Page or call ABAQUS, Inc., or your local ABAQUS representative.

Documentation

The following documentation and publications are available from ABAQUS, unless otherwise specified, in printed form and through the ABAQUS online documentation. For more information on accessing the online books, refer to the discussion of execution procedures in the ABAQUS Analysis User's Manual.

Modeling and Visualization

- **ABAQUS/CAE User's Manual:** This reference document for ABAQUS/CAE includes detailed descriptions of how to use ABAQUS/CAE for model generation, analysis, and results evaluation and visualization. ABAQUS/Viewer users should refer to the information on the Visualization module in this manual.

Analysis

- **ABAQUS Analysis User's Manual:** This volume contains a complete description of the elements, material models, procedures, input specifications, etc. It is the basic reference document for ABAQUS/Standard and ABAQUS/Explicit. Both input file usage and ABAQUS/CAE usage information are provided in this manual.

Examples

- **ABAQUS Example Problems Manual:** This volume contains more than 125 detailed examples designed to illustrate the approaches and decisions needed to perform meaningful linear and nonlinear analysis. Typical cases are large motion of an elastic-plastic pipe hitting a rigid wall; inelastic buckling collapse of a thin-walled elbow; explosive loading of an elastic, viscoplastic thin ring; consolidation under a footing; buckling of a composite shell with a hole; and deep drawing of a metal sheet. It is generally useful to look for relevant examples in this manual and to review them when embarking on a new class of problem.
- **ABAQUS Benchmarks Manual:** This online-only volume contains over 250 benchmark problems and standard analyses used to evaluate the performance of ABAQUS; the tests are multiple element tests of simple geometries or simplified versions of real problems. The NAFEMS benchmark problems are included in this manual.

Training

- **Getting Started with ABAQUS:** This document is a self-paced tutorial designed to help new users become familiar with using ABAQUS/CAE to create solid, shell, and framework models and ABAQUS/Standard or ABAQUS/Explicit to perform static, quasi-static, and dynamic stress analysis simulations. It contains a number of fully worked examples that provide practical guidelines for performing structural analyses with ABAQUS. In addition, three comprehensive tutorials are provided to introduce users familiar with the ABAQUS solver products to the ABAQUS/CAE interface.
- **Getting Started with ABAQUS/Standard: Keywords Version:** This online-only document is designed to help new users become familiar with the ABAQUS/Standard input file syntax for static and dynamic stress analysis simulations. The ABAQUS/Standard keyword interface is used to model examples similar to those included in Getting Started with ABAQUS.
- **Getting Started with ABAQUS/Explicit: Keywords Version:** This online-only document is designed to help new users become familiar with the ABAQUS/Explicit input file syntax for quasi-static and dynamic stress analysis simulations. The ABAQUS/Explicit keyword interface is used to model examples similar to those included in Getting Started with ABAQUS.
- **Lecture Notes:** These notes are available on many topics to which ABAQUS is applied. They are used in the technical seminars that ABAQUS, Inc., presents to help users improve their understanding and usage of ABAQUS (see the “Training” section above for more information about these seminars). While not intended as stand-alone tutorial material, they are sufficiently comprehensive that they can usually be used in that mode. The list of available lecture notes is included in the Documentation Price List.

Documentation Information

- **Using ABAQUS Online Documentation:** This online-only manual contains instructions for viewing and searching the ABAQUS online documentation.

Reference

- **ABAQUS Keywords Reference Manual:** This volume contains a complete description of all the input options that are available in ABAQUS/Standard and ABAQUS/Explicit.
- **ABAQUS Theory Manual:** This online-only volume contains detailed, precise discussions of all theoretical aspects of ABAQUS. It is written to be understood by users with an engineering background.
- **ABAQUS Verification Manual:** This online-only volume describes more than 12,000 basic test cases, providing verification of each individual program feature (procedures, output options, MPCs, etc.) against exact calculations and other published results. It may be useful to run these problems when learning to use a new capability. In addition, the supplied input data files provide good starting points to check the behavior of elements, materials, etc.
- **Quality Assurance Plan:** This document describes the QA procedures followed by ABAQUS. It is a controlled document, provided to customers who subscribe to either the Nuclear QA Program or the Quality Monitoring Service.

Update Information

- **ABAQUS Release Notes:** This document contains brief descriptions of the new features available in the latest release of the ABAQUS product line.

Programming

- **ABAQUS Scripting User's Manual:** This online-only manual provides a description of the ABAQUS Scripting Interface. The manual describes how commands can be used to create and analyze ABAQUS/CAE models, to view the results of the analysis, and to automate repetitive tasks. It also contains information on using the ABAQUS Scripting Interface or C++ as an application programming interface (API) to the output database.
- **ABAQUS Scripting Reference Manual:** This online-only manual provides a command reference that lists the syntax of each command in the ABAQUS Scripting Interface.
- **ABAQUS GUI Toolkit User's Manual:** This online-only manual provides a description of the ABAQUS GUI Toolkit. The manual describes the components and organization of the ABAQUS GUI. It also describes how you can customize the ABAQUS GUI to build a particular application.
- **ABAQUS GUI Toolkit Reference Manual:** This online-only manual provides a command reference that lists the syntax of each command in the ABAQUS GUI Toolkit.

Interfaces

- **ABAQUS Interface for MSC.ADAMS User's Manual:** This document describes how to use the ABAQUS Interface for MSC.ADAMS, which creates ABAQUS models of MSC.ADAMS components and converts the ABAQUS results into an MSC.ADAMS modal neutral file that can be used by the ADAMS/Flex program. It is the basic reference document for the ABAQUS Interface for MSC.ADAMS.
- **ABAQUS Interface for MOLDFLOW User's Manual:** This document describes how to use the ABAQUS Interface for MOLDFLOW, which creates a partial ABAQUS input file by translating results from a MOLDFLOW polymer processing simulation. It is the basic reference document for the ABAQUS Interface for MOLDFLOW.

Installation and Licensing

- **ABAQUS Installation and Licensing Guide:** This document describes how to install ABAQUS and how to configure the installation for particular circumstances. Some of this information, of most relevance to users, is also provided in the ABAQUS Analysis User's Manual.

CONTENTS

1. Introduction

What information does this manual contain?

What is the ABAQUS Interface for MSC.ADAMS?

What are the procedures for using the ABAQUS Interface for MSC.ADAMS?

What are the contents of the modal neutral file?

2. Preparing the ABAQUS input file

The ABAQUS substructure model

Setting up the ABAQUS model to create a modal neutral file without stress or strain

Setting up the ABAQUS model to create a modal neutral file with stress or strain

Supported ABAQUS elements

3. Creating the MSC.ADAMS modal neutral file

Units

Executing the adams command to create a modal neutral file without stress or strain

Executing the adams command to create a modal neutral file with stress or strain

 Creating the second input file 3.3.1

 Creating the modal neutral file from two results files 3.3.2

Translating modes with negative eigenvalues

Diagnosing error messages and problems

4. Examples

Example 1: Link modeled with solid elements

Example 2: Link modeled with beam elements

Example 3: Tire

1. INTRODUCTION

This chapter provides an overview of the ABAQUS Interface for MSC.ADAMS. The following topics are covered:

- “What information does this manual contain?,” Section 1.1
- “What is the ABAQUS Interface for MSC.ADAMS?,” Section 1.2
- “What are the procedures for using the ABAQUS Interface for MSC.ADAMS?,” Section 1.3
- “What are the contents of the modal neutral file?,” Section 1.4

The installation of the ABAQUS Interface for MSC.ADAMS is included in the ABAQUS product installation. For information on installing ABAQUS, see the ABAQUS Installation and Licensing Guide.

1.1 What information does this manual contain?

This manual explains how to use the ABAQUS Interface for MSC.ADAMS. For general information about using MSC.ADAMS, see the MSC.ADAMS collection of documentation. You might find the following MSC.ADAMS manuals particularly useful:

- *Using ADAMS/Flex*
- *Using ADAMS/View*

1.2 What is the ABAQUS Interface for MSC.ADAMS?

The ADAMS/Flex product from MSC.Software Corporation can be used to account for flexibility in a component when performing a dynamic analysis in MSC.ADAMS. ADAMS/Flex relies on a finite element analysis code such as ABAQUS to provide the component’s flexibility information in a form that is usable by MSC.ADAMS. The ABAQUS Interface for MSC.ADAMS can be used to create ABAQUS models of MSC.ADAMS components and to convert the ABAQUS results into an MSC.ADAMS modal neutral (**.mnf**) file, the format required by ADAMS/Flex.

The ABAQUS Interface for MSC.ADAMS Version 6.5 requires a results file created by ABAQUS Version 6.2 or later. The ABAQUS Interface for MSC.ADAMS creates modal neutral files compatible with ADAMS Version 10.1 and later. Modal stress and strain, if present in the modal neutral file, require ADAMS Version 12 or later.

1.3 What are the procedures for using the ABAQUS Interface for MSC.ADAMS?

The typical usage of the ABAQUS Interface for MSC.ADAMS involves creating a modal neutral file without stress or strain from a single results file and requires one ABAQUS analysis and one ABAQUS Interface for MSC.ADAMS step. The following procedure summarizes the typical usage of the ABAQUS Interface for MSC.ADAMS:

To use the ABAQUS Interface for MSC.ADAMS:

1. Create an ABAQUS model for each flexible component of the MSC.ADAMS model. Each component is modeled as an ABAQUS substructure.
2. Run the ABAQUS analysis.
3. Run the ABAQUS Interface for MSC.ADAMS to read the ABAQUS results file produced by the analysis and to create the modal neutral (**.mnf**) file for MSC.ADAMS.
4. Read the modal neutral file into MSC.ADAMS. A separate modal neutral file must be created for each flexible part in MSC.ADAMS.

If you want the ABAQUS Interface for MSC.ADAMS to translate stress or strain to the modal neutral file, you can modify the general procedure. In the modified procedure the ABAQUS Interface for MSC.ADAMS creates a modal neutral file from two results files and requires two ABAQUS analyses and two ABAQUS Interface for MSC.ADAMS steps. For more information, see “Setting up the ABAQUS model to create a modal neutral file with stress or strain,” Section 2.3, and “Executing the **adams** command to create a modal neutral file with stress or strain,” Section 3.3.

The remaining sections of this manual discuss these procedures in detail.

1.4 What are the contents of the modal neutral file?

The ABAQUS Interface for MSC.ADAMS translates data from one or more ABAQUS results (**.fil**) files and creates an MSC.ADAMS modal neutral (**.mnf**) file. Depending on the contents of the results files and the translation parameters, the ABAQUS Interface for MSC.ADAMS creates a modal neutral file containing the data blocks shown in Table 1–1.

Table 1–1 Modal neutral file contents.

Block Number	Contents	Created by the ABAQUS Interface for MSC.ADAMS
1	Version code	Yes
2	Header	Yes
3	Content summary	Yes
4	Nodal coordinates	Yes
5	<Not used>	N/A
6	Global mass properties	Yes
7	Eigenvalues	Yes
8	Mode shapes	Yes
9	Nodal masses	Yes
10	Nodal inertias	Yes
11	Units	Yes
12	Generalized stiffness matrix	Yes
13	Generalized mass matrix	Yes
14	Element faces	Yes
15	Generalized damping	No
16	Mode shape transformation	Yes
17	Interface nodes	Yes
18	Modal stress	Optional
19 to 26	Inertia invariants	Yes
27	Modal preload	Yes
28	Modal loads	No
29	Modal strain	Optional

2. PREPARING THE ABAQUS INPUT FILE

This chapter describes the preparation of an ABAQUS input file that will produce the results quantities required by ADAMS/Flex.

2.1 The ABAQUS substructure model

The first step in accounting for a component’s flexibility in MSC.ADAMS is to model that component as an ABAQUS substructure. This process involves creating an ABAQUS finite element model of the component. General guidelines for building ABAQUS models with substructures are described in “Using substructures,” Section 7.2.1 of the ABAQUS Analysis User’s Manual. The specific requirements for building substructure models that can be exported to MSC.ADAMS are described in the following sections.

The ABAQUS Interface for MSC.ADAMS creates modal neutral files that do not contain stress or strain from a single results file that is written by an ABAQUS analysis, as described in “Setting up the ABAQUS model to create a modal neutral file without stress or strain,” Section 2.2. If you want the ABAQUS Interface for MSC.ADAMS to translate stress or strain to the modal neutral file, a second ABAQUS analysis is required, as discussed in “Setting up the ABAQUS model to create a modal neutral file with stress or strain,” Section 2.3.

2.2 Setting up the ABAQUS model to create a modal neutral file without stress or strain

If you want the ABAQUS Interface for MSC.ADAMS to create a modal neutral file without stress or strain, you can use the following template to prepare an input file for the ABAQUS analysis:

```

*HEADING
. . .
*****
*STEP
*FREQUENCY, EIGENSOLVER=...
. . .
*BOUNDARY
. . .
*ELEMENT MATRIX OUTPUT, MASS=YES, ELSET=...
*NODE FILE
U,
*END STEP
*****
    
```

PREPARING THE ABAQUS INPUT FILE

```
*STEP, UNSYMM=NO
*SUBSTRUCTURE GENERATE, TYPE=Z...,
  RECOVERY MATRIX=YES, MASS MATRIX=YES
*RETAINED NODAL DOFS
. . .
*RETAINED EIGENMODES
. . .
*SUBSTRUCTURE LOAD CASE, NAME=...
*CLOAD
. . .
*SUBSTRUCTURE MATRIX OUTPUT,
  RECOVERY MATRIX=YES, MASS=YES,
  STIFFNESS=YES, SLOAD=YES
*END STEP
*****
```

The history section of the input file must contain a *FREQUENCY step to calculate the fixed-interface normal modes, followed by a *SUBSTRUCTURE GENERATE step. The *FREQUENCY step may be preceded by any number of steps to apply a desired preload to the model.

Note the following points about the *FREQUENCY step:

- The *FREQUENCY step must apply zero boundary conditions to every degree of freedom that will be retained in the *SUBSTRUCTURE GENERATE step. Other degrees of freedom may be constrained as appropriate.
- This step must write element mass matrices and eigenvectors to the results file.

Note the following points about the *SUBSTRUCTURE GENERATE step:

- The UNSYMM=NO parameter on the *STEP option is optional but recommended. Certain preloading histories (for example, contact with high friction coefficients) may create unsymmetric stiffness matrices. The substructure matrix created after such a preloading history will in all cases be symmetric. However, by default, ABAQUS will create and write the full substructure matrix to the results file if the stiffness matrix was unsymmetric earlier in the analysis. Using UNSYMM=NO saves computation time and storage space without affecting accuracy.
- The *SUBSTRUCTURE GENERATE option must include the parameters RECOVERY MATRIX=YES and MASS MATRIX=YES.
- The list of *RETAINED NODAL DOFS must be equal to or a subset of the list of DOFs set to zero in the preceding *FREQUENCY step.
- The list of *RETAINED EIGENMODES must be equal to or a subset of the eigenmodes computed in the *FREQUENCY step. If the list is a subset, unused eigenmodes must not be written to the results file.
- The *SUBSTRUCTURE MATRIX OUTPUT option must write the recovery matrix, stiffness matrix, and mass matrix to the results file.

- If the SLOAD=YES parameter is used on the *SUBSTRUCTURE MATRIX OUTPUT option, modal load components corresponding to all internal and external loads acting on the substructure will be translated to modal preloads (block 27) in the modal neutral file.
- The *SUBSTRUCTURE LOAD CASE option is optional. If the option is present, *CLOAD data should duplicate the loading defined in an earlier general step to represent the effect of other parts of the model on the substructure. As noted in the preceding paragraph, if SLOAD=YES is used on the *SUBSTRUCTURE MATRIX OUTPUT option, these external loads (along with any internal loads) will be written to the preload data block of the modal neutral file. The NAME used for the load case is immaterial.

The history section of the ABAQUS input file may include general steps preceding the required *FREQUENCY and *SUBSTRUCTURE GENERATE steps. Note the following points about these optional general steps:

- Displacements written to the results file in these steps will be used to update the nodal coordinates written to the modal neutral file. Displacements for later steps will update those of earlier steps. Therefore, if displacements for a subset of nodes have been written for any step, the final step should write displacements for all nodes of the substructure; otherwise, some nodes will be translated with their original coordinates, and others will be translated with their deformed coordinates.
- Loads from these steps may be translated to modal preloads (block 27) in the modal neutral file if SLOAD=YES is used on the *SUBSTRUCTURE MATRIX OUTPUT option. Internal and external loads are treated differently:
 - Internal loads include distributed loads (such as self-weight) and boundary conditions. These loads are considered to be an intrinsic part of the substructure. All *DLOAD data and the reaction forces due to boundary conditions are treated as internal loads.

The resultant of internal loads may be nonzero. For example, if a gravity load is applied to a structure restrained from rigid body motion, a net force equal to the weight of the body will act on the rest of the MSC.ADAMS model through the retained degrees of freedom.
 - External loads represent the effect of other parts of the model on the substructure. All *CLOAD options are considered to be external loads. To translate these loads to MSC.ADAMS, the *CLOADs in a general step must be replicated as *SUBSTRUCTURE LOAD CASE data in the *SUBSTRUCTURE GENERATE step.

2.3 Setting up the ABAQUS model to create a modal neutral file with stress or strain

If you want the ABAQUS Interface for MSC.ADAMS to translate stress or strain to the modal neutral file, you must modify the template in the previous section to prepare an input file for the first ABAQUS analysis. You must include an output request for stress or strain in the *FREQUENCY step, as shown in the following example:

PREPARING THE ABAQUS INPUT FILE

```
*****  
*EL FILE, POSITION=NODES, DIRECTIONS=YES  
1,  
S,  
E,  
*****
```

Note the following points about the output request:

- The POSITION=NODES parameter is required.
- The DIRECTIONS=YES parameter is recommended for all models. This parameter is required for models containing shell elements.
- The section point number (**1**, on the line following ***EL FILE**, in this example) is required for models containing shell elements. The section point number will be ignored for solid elements. Stress or strain for only a single section point can be translated to the modal neutral file.
- The output variables stress (**S**), strain (**E**), or both can be written to the results file.

In addition, you must run a second ABAQUS analysis to recover stress or strain in the substructure for the static constraint modes. You can automatically create the input file for the second ABAQUS analysis using the procedure described in “Creating the second input file,” Section 3.3.1.

2.4 Supported ABAQUS elements

The ABAQUS Interface for MSC.ADAMS is designed to support most ABAQUS elements that have displacement degrees of freedom at any node. However, there are a few limitations and considerations. Infinite elements (for example, CIN3D8), coupled thermal-structural elements (for example, C3D8T), generalized axisymmetric elements (for example, CGAX3), and frame elements (for example, FRAME3D) are not supported.

3. CREATING THE MSC.ADAMS MODAL NEUTRAL FILE

This chapter describes the procedures used to create the MSC.ADAMS modal neutral (.**mnf**) file from the ABAQUS results files. The procedure varies depending on whether stress or strain are to be translated to the modal neutral file, as described in “Executing the **adams** command to create a modal neutral file without stress or strain,” Section 3.2, and “Executing the **adams** command to create a modal neutral file with stress or strain,” Section 3.3.

3.1 Units

The MSC.ADAMS programs require that the user define the units used in the component model, while ABAQUS does not. Therefore, during the creation of the modal neutral file the user must declare explicitly the units used in the model. The approach to doing this in the ABAQUS Interface for MSC.ADAMS is very similar to the way it is done in the ADAMS/View **Units Settings** dialog box. A predefined units system can be specified by using the **units** option on the ABAQUS Interface for MSC.ADAMS execution procedure. Alternatively, the individual length, mass, force, and time units can be specified by using the **length**, **mass**, **force**, and **time** options on the ABAQUS Interface for MSC.ADAMS execution procedure. Any individual units that are specified override the corresponding units in the units system. The default units system is **mks**. The valid units systems for the **units** option are listed in Table 3–1.

Table 3–1 Valid units systems.

Units System	Length Units	Mass Units	Force Units	Time Units
mks	meters	kilograms	Newtons	seconds
mmks	millimeters	kilograms	Newtons	seconds
cgs	centimeters	grams	dyne	seconds
ips	inches	slugs	pound-force	seconds

The valid options for each of the **length**, **mass**, **force**, and **time** options are as follows:

Length units

Valid options for the length units are

- **meters**
- **millimeters, mm**
- **centimeters, cm**
- **kilometers, km**
- **inches, inch, in**

CREATING THE MSC.ADAMS MODAL NEUTRAL FILE

- **feet, foot, ft**
- **mile**

Mass units

Valid options for the mass units are

- **kilograms, kg**
- **megagram, tonne**
- **gram, g**
- **pound_mass, lbm, pound**
- **slug**
- **kpound_mass**
- **ounce_mass**

Force units

Valid options for the force units are

- **Newtons, N**
- **kNewton, kN**
- **kilogram_force, kgf**
- **dyne**
- **ounce_force**
- **pound_force, lbf, pound**
- **kpound_force**

Time units

Valid options for the time units are

- **seconds, sec**
- **milliseconds, ms**
- **minutes, min**
- **hours**

3.2 Executing the adams command to create a modal neutral file without stress or strain

The **adams** command is used to read the ABAQUS results file produced by the multi-step ABAQUS analysis and to produce an MSC.ADAMS modal neutral (**.mnf**) file. There are several specific requirements on the format of the results file. Creating an input file to satisfy these requirements is

described in “Setting up the ABAQUS model to create a modal neutral file without stress or strain,” Section 2.2.

```
abaqus adams           job=job-name
                        [input=input-file]
                        [units=mmks | mks | cgs | ips]
                        [length=length-units-name]
                        [mass=mass-units-name]
                        [time=time-units-name]
                        [force=force-units-name]
```

job

This option specifies the input and output file names to use during results file translation. The *job-name* value is used to construct the default input file name, *job-name.fil*. The output modal neutral file is given the name *job-name.mnf*.

If this option is omitted from the command line, the user will be prompted for this value.

input

This option specifies the name of the results file if it is different from *job-name.fil*.

units

This option specifies the units system for the model. The possible values are **mmks**, **mks**, **cgs**, or **ips**, which correspond to the ADAMS/View options with the same names. The default value is **mks**.

length

This option specifies the length units for the model. If this option is specified, it overrides the length units of the specified units system.

mass

This option specifies the mass units for the model. If this option is specified, it overrides the mass units of the specified units system.

time

This option specifies the time units for the model. If this option is specified, it overrides the time units of the specified units system.

force

This option specifies the force units for the model. If this option is specified, it overrides the force units of the specified units system.

CREATING THE MSC.ADAMS MODAL NEUTRAL FILE

Default values for the units options can be defined in the ABAQUS environment file (**abaqus_v6.env**). The default for the **units** option can be defined with the **adams_units_family** parameter. The defaults for the **length**, **mass**, **time**, and **force** options can be defined with the **adams_length_units**, **adams_mass_units**, **adams_time_units**, and **adams_force_units** parameters, respectively.

3.3 Executing the adams command to create a modal neutral file with stress or strain

To create a modal neutral file containing stress or strain, the ABAQUS Interface for MSC.ADAMS must read data from two results files. You create the first results file according to the requirements described in “Setting up the ABAQUS model to create a modal neutral file with stress or strain,” Section 2.3. Next, you use the **adams** command to create an input file for the second ABAQUS analysis as described in “Creating the second input file,” Section 3.3.1. You then run the second ABAQUS analysis, which writes the second results file. Finally, you use the **adams** command to create the modal neutral file as described in “Creating the modal neutral file from two results files,” Section 3.3.2.

3.3.1 Creating the second input file

To create the input file for the second ABAQUS analysis, execute the ABAQUS Interface for MSC.ADAMS using the following command:

```
abaqus adams           job=job-name  
                        [input=input-file]  
                        [make_se_recovery]  
                        [stress_modes={ON | OFF}]  
                        [strain_modes={ON | OFF}]  
                        [section_point=section_point_number]
```

job

This option controls the name of the new input file that will be created. The *job-name* value is used to construct the new input file name, *job-name_se_recovery.inp*.

If this option is omitted from the command line, the user will be prompted for this value.

input

This option specifies the name of the results file from the first ABAQUS analysis, if it is different from *job-name.fil*.

make_se_recovery

This option specifies that the translator is to create a new ABAQUS input file. A modal neutral file will not be created.

stress_modes

This option specifies that the new ABAQUS input file will contain commands to write stress to the results file. The possible values are ON and OFF. The default value is ON.

strain_modes

This option specifies that the new ABAQUS input file will contain commands to write strain to the results file. The possible values are ON and OFF. The default value is OFF.

section_point

This option specifies the section point at which shell stresses and/or strains will be written to the results file. The default value is 1. This option will be ignored for three-dimensional continuum elements.

3.3.2 Creating the modal neutral file from two results files

After running the two ABAQUS analyses, as described above, you create a modal neutral file containing modal stress or strain by executing the ABAQUS Interface for MSC.ADAMS using the following command:

```
abaqus adams           job=job-name
                        [input=input-file]
                        [se_recovery_job=se_recovery_job-name]
                        [units=mmks | mks | cgs | ips]
                        [length=length-units-name]
                        [mass=mass-units-name]
                        [time=time-units-name]
                        [force=force-units-name]
```

job

This option controls the name of the modal neutral file that will be created. The *job-name* value is used to construct the new modal neutral file name, *job-name.mnf*.

If this option is omitted from the command line, the user will be prompted for this value.

input

This option specifies the name of the results file from the first ABAQUS analysis, if it is different from *job-name.fil*.

se_recovery_job

This option specifies the name of the results file from the second ABAQUS analysis.

units

This option specifies the units system for the model. The possible values are **mmks**, **mks**, **cgs**, or **ips**, which correspond to the ADAMS/View options with the same names. The default value is **mks**.

length

This option specifies the length units for the model. If this option is specified, it overrides the length units of the specified units system.

mass

This option specifies the mass units for the model. If this option is specified, it overrides the mass units of the specified units system.

time

This option specifies the time units for the model. If this option is specified, it overrides the time units of the specified units system.

force

This option specifies the force units for the model. If this option is specified, it overrides the force units of the specified units system.

3.4 Translating modes with negative eigenvalues

The ABAQUS Interface for MSC.ADAMS uses component modal synthesis to combine the fixed-interface normal modes and the substructure recovery vectors into a basis of modal degrees of freedom that will be used for dynamic analysis in MSC.ADAMS. This modal basis spans a space that includes the rigid body response of the substructure. Typically, for a non-prestressed, unrestrained body in three dimensions, one expects to find six rigid body modes with associated zero eigenvalues.

The situation is, in general, different for prestressed models, where an unrestrained structure may have less than six modes with zero eigenvalues. Prestressing may change the expected zeroes into values that are significantly positive or negative, depending on the sign of the prestress.

By default, the ABAQUS Interface for MSC.ADAMS deletes modes with negative eigenvalues and reorthogonalizes the reduced basis. If you want to retain modes with negative eigenvalues, define the environment variable.

- On UNIX platforms type the following command:

```
setenv MDI_MNFWRITE_OPTIONS negative_roots_OK
```

- On Windows platforms type the following command:

```
set MDI_MNFWRITE_OPTIONS=negative_roots_OK
```

In this case the ABAQUS Interface for MSC.ADAMS will treat modes with negative eigenvalues in the same manner as all other modes.

To determine if a model will have negative eigenvalues when translated by the ABAQUS Interface for MSC.ADAMS, you can add a *FREQUENCY step with no boundary conditions to the input file. If this step is added to the run that creates the results file used by the ABAQUS Interface for MSC.ADAMS, it must not write anything to the results file.

3.5 Diagnosing error messages and problems

During execution of the **adams** command, the following warning and error messages may be output:

```
WARNING: There are 'N' elements in substructure Z1, but only 'M'  
-=> mass matrices have been processed from the results file.  
-=> Carefully review this discrepancy before proceeding.
```

Explanation: If the number of mass matrices read is zero, verify that there is an *ELEMENT MATRIX OUTPUT, MASS=YES option in the *FREQUENCY step that preceded the *SUBSTRUCTURE GENERATE step.

If m is nonzero but less than n , the model may be correct. Some ABAQUS elements, such as dashpots, do not have mass matrices. The mass of other elements may be neglected if they are not significant in representing the mass of the substructure. Elements with negligible mass are not required in the element set whose mass is written to the results file.

```
WARNING: No fixed-interface normal modes.  
-=> The results file did not contain any modes from  
-=> a *FREQUENCY step. Typically, this step contains:  
-=> *NODE FILE  
-=> U,
```

Explanation: Vibration mode shapes were missing in the results file. A *FREQUENCY step must precede the *SUBSTRUCTURE GENERATE step. Vibration mode shapes from the *FREQUENCY step are written to the results file using the commands:

```
*NODE FILE  
U,
```

Typically, all the mode shapes will be written. However, if a subset of the computed modes is retained in the *SUBSTRUCTURE GENERATE step (using the *RETAINED EIGENMODES option), the *FREQUENCY step must write only those modes.

CREATING THE MSC.ADAMS MODAL NEUTRAL FILE

ERROR: Missing element mass matrices.

- => No element mass matrices were found in the results file.
- => The input file must contain an entry similar to:
- => *ELEMENT MATRIX OUTPUT, MASS=YES, ELSET=elset_name
- => This entry must be in the *FREQUENCY step preceding
- => the *SUBSTRUCTURE GENERATE step.

Explanation: The *FREQUENCY step must write element mass matrices to the results file using the syntax given in the error message.

ERROR: Missing generalized stiffness and mass matrices.

- => Verify that the input file defines this substructure
- => and contains the following option:
- => *SUBSTRUCTURE MATRIX OUTPUT, STIFFNESS=YES, MASS=YES,
- => RECOVERY MATRIX=YES [, SLOAD=YES]

Explanation: The *SUBSTRUCTURE GENERATE step must write the generalized (reduced) mass and stiffness matrices, as well as the recovery matrices, to the results file. The substructure load vector may also be written using SLOAD=YES.

ERROR: Deprecated superelement data in the results file.

- => ABAQUS/ADAMS Version 6.2 and later require a results file
- => created using the SUBSTRUCTURE keywords introduced
- => in ABAQUS Version 6.2.
- => The ABAQUS analysis must be rerun.

Explanation: Results files created with ABAQUS Version 6.1 or earlier cannot be translated by ABAQUS/ADAMS Version 6.2 and later.

4. EXAMPLES

This chapter contains three example problems. The first two examples model a simple flexible link component and can be used in the tutorials and examples in the MSC.ADAMS document *Using ADAMS/Flex*. The third example is a tire that is prestressed by inflation and contact with the road prior to creating the ABAQUS substructure.

4.1 Example 1: Link modeled with solid elements

This example models a simple flexible link component using three-dimensional continuum elements.

To perform the analysis for the link modeled with solid elements:

1. Enter the following commands to extract the input files from the compressed archive files provided with the ABAQUS release:

```
abaqus fetch job=adams_ex1
abaqus fetch job=adams_ex1_nodes
abaqus fetch job=adams_ex1_elements
```

2. Enter the following command to execute the ABAQUS analysis:

```
abaqus job=adams_ex1
```

3. Enter the following command to execute the ABAQUS Interface for MSC.ADAMS and translate the results file generated in the ABAQUS analysis to a modal neutral file for use with ADAMS/Flex:

```
abaqus adams job=adams_ex1
```

The solid element link model used in the MSC.ADAMS four-bar linkage model is shown in Figure 4–1. The link is modeled with 642 C3D10 tetrahedral solid elements (1368 nodes).

Because the solid elements have only displacement degrees of freedom at their nodes, multi-point constraints are used to provide a connection to the other components in the MSC.ADAMS model. Two nodes are added along the centerline of the beam at the centers of the hinge holes. The C3D10 nodes that lie on the faces of the hinge holes are connected to the extra nodes with BEAM-type multi-point constraints, allowing the nodes to transmit both forces and moments between the link and other MSC.ADAMS components.

The options used to define the single substructure are those described in “The ABAQUS substructure model,” Section 2.1. Twenty fixed-interface vibration modes are computed to represent the dynamic behavior of the link.

MSC.ADAMS uses the fixed-interface vibration modes and the constraint modes to characterize the flexibility of the link. The eight lowest fixed-interface vibration frequencies computed by ABAQUS are shown in Table 4–1. These frequencies are reported in the **adams_ex1.dat** file.

EXAMPLES

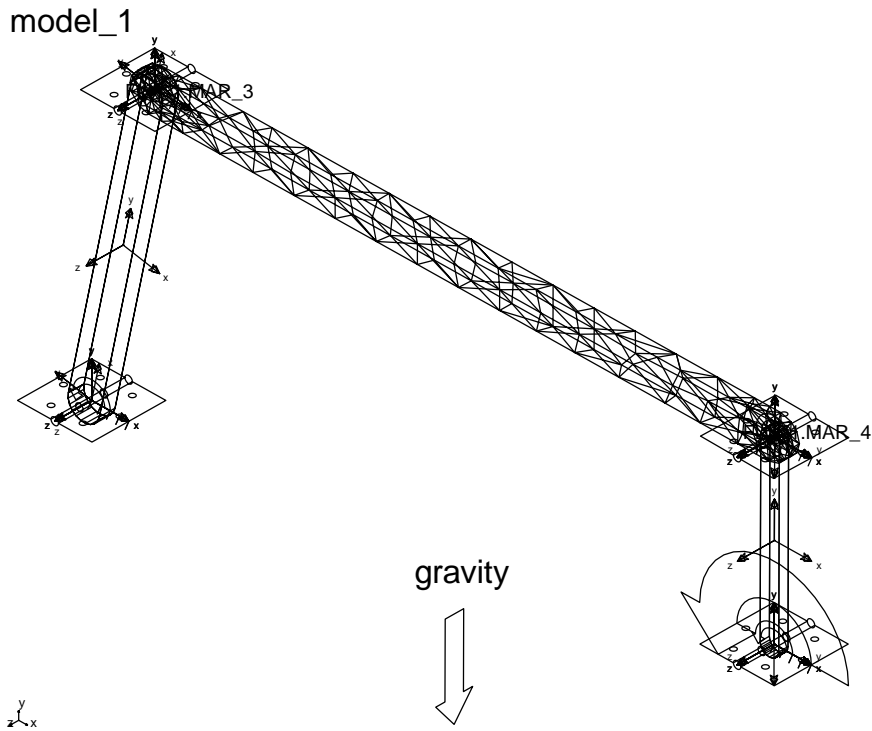


Figure 4-1 Solid link model.

Table 4-1 Fixed-interface vibration frequencies for the solid link model.

Frequency, Hz
206
391
570
1124
1228
1817
1879
2541

The ABAQUS Interface for MSC.ADAMS combines these fixed-interface modes with the static constraint modes to compute an equivalent modal basis to be used by ADAMS/Flex. The first six frequencies of this equivalent basis are approximately zero. The next eight frequencies for the unconstrained model are shown in Table 4–2. These frequencies are written to the screen when executing the ABAQUS Interface for MSC.ADAMS.

Table 4–2 Nonzero frequencies for the solid link model that are used by ADAMS/Flex.

Frequency, Hz
194
535
574
1055
1551
1762
1801
2653

The ABAQUS input file for the solid model, **adams_ex1.inp**, is shown below.

```

*HEADING
Link modeled with C3D10 solid elements
** -----
**
**               NODE DEFINITION
**
** *NODE,input=adams_ex1_nodes.inp
**
**
** *NSET, NSET=LEFTCYL
8,9,17,18,70,71,72,73,125,126,127,128,134,135,207,
229,230,278,309,310,311,312,313,314,373,374,375,376,
377,378,389,390,391,392,498,533,534,535,546,565,566,
677,688,734,1058,1059,1073,1085,1114,1115,1311,1312,
1325,1335,1356,1357
**
** *NSET, NSET=RIGHTCYL
6,7,15,16,66,67,68,69,121,122,123,124,136,137,231,

```

EXAMPLES

```
232,303,304,305,306,307,308,367,368,369,370,371,372,
393,394,395,396,479,480,481,487,488,506,635,654,957,
958,976,977,1004,1026,1219,1220,1234,1235,1257,1287
**
*MPC
BEAM,LEFTCYL,10000
BEAM,RIGHTCYL,20000
** -----
**
**                      ELEMENT DEFINITION
**
*ELEMENT,TYPE=C3D10,ELSET=PROP1,INPUT=adams_ex1_elements.inp
**
** -----
**
**                      ELEMENT PROPERTY DEFINITION
**
**
*SOLID SECTION,ELSET=PROP1,MATERIAL=STEEL
**
** -----
**
**                      MATERIAL DEFINITION
**
**
*MATERIAL,NAME=STEEL
*ELASTIC
  2.069999944E+11, 3.000000119E-01,
*DENSITY
  7.800000000E+03,
**
*NSET,NSET=RETNODES
10000,20000
**
** -----
**
*STEP
*FREQUENCY,EIGENSOLVER=LANCZOS
20,
*BOUNDARY
RETNODES, 1,6
*ELEMENT MATRIX OUTPUT, MASS=YES, ELSET=PROP1
```

```

*NODE FILE
U
*END STEP
**
** -----
**
**              SUBSTRUCTURE GENERATION
**
*STEP
*SUBSTRUCTURE GENERATE, TYPE=Z1, RECOVERY MATRIX=YES,
  MASS MATRIX=YES, OVERWRITE
*RETAINED NODAL DOFS, SORTED=NO
RETNODES, 1, 6
*RETAINED EIGENMODES, GENERATE
1, 20
*SUBSTRUCTURE MATRIX OUTPUT, STIFFNESS=YES, MASS=YES,
  RECOVERY=YES
*END STEP

```

4.2 Example 2: Link modeled with beam elements

This example models a simple flexible link component using three-dimensional beam elements.

To perform the analysis for the link modeled with beam elements:

1. Enter the following command to extract the input files from the compressed archive files provided with the ABAQUS release:

```
abaqus fetch job=adams_ex2
```

2. Enter the following command to execute the ABAQUS analysis:

```
abaqus job=adams_ex2
```

3. Enter the following command to execute the ABAQUS Interface for MSC.ADAMS and translate the results file generated in the ABAQUS analysis to a modal neutral file for use with ADAMS/Flex:

```
abaqus adams job=adams_ex2
```

The primary difference between the beam model and the solid model is that the beam model uses only 10 B31 elements (11 nodes). Because the beam elements have both displacement and rotational degrees of freedom at their nodes, no multi-point constraints are needed to connect the link to other MSC.ADAMS components. The rest of the model is essentially identical to the solid model of the link.

EXAMPLES

The first eight nonzero frequencies for the unconstrained model are shown in Table 4–3.

Table 4–3 Nonzero frequencies for the beam link model that are used by ADAMS/Flex.

Frequency, Hz
205
555
610
1070
1618
1742
1775
2568

These frequencies are close to those of the solid model of the link. Although the computational cost in ABAQUS is much less for this model than for the solid model, the computational costs in MSC.ADAMS for the two models are very similar because both models have 32 modes (12 constraint modes and 20 fixed-interface vibration modes).

The ABAQUS input file for the beam model, `adams_ex2.inp`, is shown below.

```
*HEADING
Link modeled with B31 beam elements
** -----
**
**                               NODE DEFINITION
**
** *NODE, nset=na11
**
    1,0.000000000E+00,0.000000000E+00,0.000000000E+00
    2,5.000000000E-02,0.000000000E+00,0.000000000E+00
    3,1.000000000E-01,0.000000000E+00,0.000000000E+00
    4,1.500000000E-01,0.000000000E+00,0.000000000E+00
    5,2.000000000E-01,0.000000000E+00,0.000000000E+00
    6,2.500000000E-01,0.000000000E+00,0.000000000E+00
    7,3.000000000E-01,0.000000000E+00,0.000000000E+00
    8,3.500000000E-01,0.000000000E+00,0.000000000E+00
    9,4.000000000E-01,0.000000000E+00,0.000000000E+00
   10,4.500000000E-01,0.000000000E+00,0.000000000E+00
   11,5.000000000E-01,0.000000000E+00,0.000000000E+00
```

```

**
** -----
**
**                ELEMENT DEFINITION
**
**
**ELEMENT,TYPE=B31
**      1,1,2
**      2,2,3
**      3,3,4
**      4,4,5
**      5,5,6
**      6,6,7
**      7,7,8
**      8,8,9
**      9,9,10
**     10,10,11
**
** -----
**
**                ELEMENT PROPERTY DEFINITION
**
**
**ELSET,ELSET=PROP1
**1,2,3,4,5,6,7,8,9,10
**
**BEAM SECTION,ELSET=PROP1,SECTION=RECT,MATERIAL=STEEL,TEMP=GRAD
**3.000E-02,1.000E-02
**0.000E+00,0.000E+00,-1.000E+00
**
**
** -----
**
**                MATERIAL DEFINITION
**
**
**MATERIAL,NAME=STEEL
**ELASTIC
**  2.069999944E+11, 3.000000119E-01,
**DENSITY
**  7.800000000E+03,
**
**
**NSET,NSET=RETNODES
**1, 11

```

EXAMPLES

```
**
** -----
**
**STEP
**FREQUENCY,EIGENSOLVER=LANCZOS
20,
**BOUNDARY
RETNODES, 1,6
**ELEMENT MATRIX OUTPUT, MASS=YES, ELSET=PROP1
**NODE FILE
U
**END STEP
**
** -----
**
**              SUBSTRUCTURE GENERATION
**
**STEP
**SUBSTRUCTURE GENERATE, TYPE=Z1, RECOVERY MATRIX=YES,
  MASS MATRIX=YES, OVERWRITE
**RETAINED NODAL DOFS, SORTED=NO
RETNODES, 1,6
**RETAINED EIGENMODES, GENERATE
1,20
**SUBSTRUCTURE MATRIX OUTPUT, STIFFNESS=YES, MASS=YES,
  RECOVERY=YES
**END STEP
```

4.3 Example 3: Tire

This example models a tire. The substructure is created after solving a highly nonlinear prestress problem to account for inflating the tire and giving it a footprint due to contact with the road.

To perform the analysis for the tire:

1. Enter the following commands to extract the input files from the compressed archive files provided with the ABAQUS release:

```
abaqus fetch job=adams_ex3A
abaqus fetch job=adams_ex3A_nodes
```

```
abaqus fetch job=adams_ex3B
abaqus fetch job=adams_ex3C
```

2. You must perform three ABAQUS analyses.
 - a. Enter the following command to solve an axisymmetric model for the tire inflation:

```
abaqus job=adams_ex3A
```

- b. Enter the following command to create the three-dimensional model of the tire from the axisymmetric model and its results and to calculate the footprint of the tire in contact with the road:

```
abaqus job=adams_ex3B oldjob=adams_ex3A
```

- c. Enter the following command to create the substructure model:

```
abaqus job=adams_ex3C oldjob=adams_ex3B
```

3. Enter commands to execute the ABAQUS Interface for MSC.ADAMS and to create a modal neutral file for use with ADAMS/Flex.

- On UNIX platforms enter the following commands:

```
setenv MDI_MNFWRITE_OPTIONS negative_roots_OK
abaqus adams job=adams_ex3C
unsetenv MDI_MNFWRITE_OPTIONS
```

- On Windows platforms enter the following commands:

```
set MDI_MNFWRITE_OPTIONS=negative_roots_OK
abaqus adams job=adams_ex3C
set MDI_MNFWRITE_OPTIONS=
```

This example extends the discussion of the model described in “Symmetric results transfer for a static tire analysis,” Section 3.1.1 of the ABAQUS Example Problems Manual. The ABAQUS analyses of **adams_ex3A** and **adams_ex3B** essentially replicate the inflation and footprint analysis of the tire as described in that section. However, a few modifications have been made to **adams_ex3B** to prepare it for the substructure analysis that follows:

- The rim and hub are modeled as a rigid body, whose reference node is located at the axle. Six degrees of freedom of the reference node will be among the retained degrees of freedom of the substructure.
- The footprint analysis is controlled by applying loads and boundary conditions to this reference node.
- The *MODEL CHANGE, ACTIVATE option is used in the first step of the analysis. This option does not affect the results of that step but is required so that the road-tire contact pair can be removed before creating the substructure in the ABAQUS restart analysis of **adams_ex3C**.

EXAMPLES

- The third step has output requests for CDISP and CSTRESS to determine the tire nodes in contact with the road at the end of the footprint analysis. A subset of these nodes will be among the retained nodes of the substructure.

The tire model in its original and deformed states is shown in Figure 4–2 and Figure 4–3.

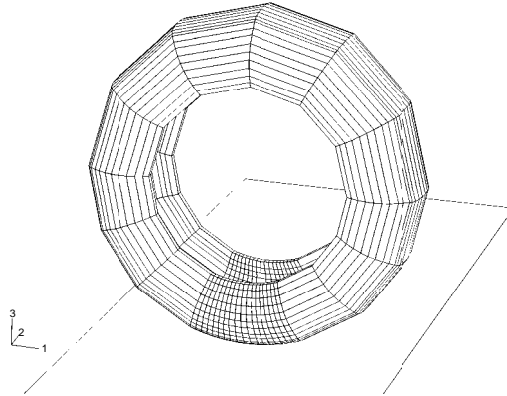


Figure 4–2 Tire model in the original state.

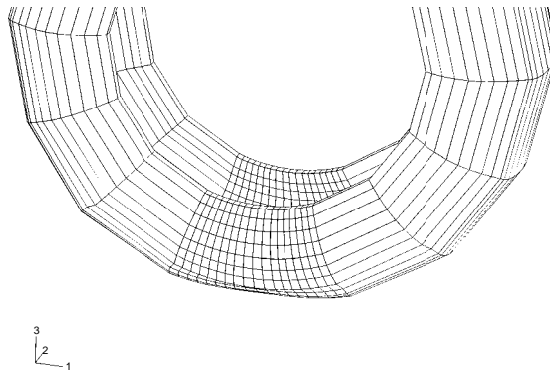


Figure 4–3 Tire model in the deformed state.

The ABAQUS analysis of **adams_ex3C** restarts from the inflation and footprint analysis of **adams_ex3B** and consists of the following three steps:

- The tire is isolated from the road. The *MODEL CHANGE, REMOVE, TYPE=CONTACT PAIR option is used to remove the rigid surface representing the road. The mechanics of the solution are

unchanged, since the *BOUNDARY, FIXED option is used to specify that the nodes in node set **FOOTPR** have displacements identical to their computed values at the end of the previous step.

One effect of this step is to reformulate the stiffness matrix of the tire without the Lagrange multipliers that were used to enforce the contact constraints; this leads to a more realistic substructure matrix.

This step writes displacements for all nodes to the results file so that deformed nodal coordinates will be written to the results file.

- Twenty normal modes of the tire are computed. This step has boundary conditions to restrain all degrees of freedom that will be retained in the substructure, plus additional restraints to maintain the footprint shape.

This step writes element mass matrices for all elements and eigenvectors for all modes to the results file. The eight lowest vibration frequencies computed in this step are shown in Table 4–4.

Table 4–4 Fixed-interface vibration frequencies for the prestressed tire.

Frequency, Hz
57
65
70
83
94
99
108
118

To compute the modes and frequencies for the unrestrained, prestressed tire, remove all boundary conditions and run a separate analysis. The eight lowest eigenvalues for this analysis are shown in Table 4–5. The prestress has eliminated four of the zero eigenvalues that would be expected in an unstressed free vibration calculation. Two of these eigenvalues are significantly negative; hence, their retention in the modal neutral file is optional and is controlled by an environment variable, as discussed in “Translating modes with negative eigenvalues,” Section 3.4.

- The substructure is created. The list of retained nodal degrees of freedom includes six degrees of freedom at the hub and three degrees of freedom at 35 nodes of the footprint. These contribute 111 degrees of freedom to the substructure. In addition, 20 fixed-interface normal modes are retained, so the substructure mass and stiffness matrices have 131 degrees of freedom. Depending on the engineering use of the substructure, you can choose other retained degrees of freedom. You can experiment with retaining a different number of nodes or possibly only the normal component of displacement at some nodes. In addition, the number of fixed-interface normal modes can be varied.

EXAMPLES

Table 4–5 Eigenvalues computed by ABAQUS for the unrestrained prestressed tire, using all DOFs of the FEA model.

Eigenvalue
-3743
-1970
0
0
0
0
3.048E+05
3.208E+05

The *SUBSTRUCTURE MATRIX OUTPUT option uses the optional parameter SLOAD=YES to write the modal load components to the results file. Thus, after translation, the loads corresponding to the fraction of vehicle weight that prestressed the tire will be in the modal neutral file used by ADAMS/Flex.

After reorthogonalizing the component modes computed by ABAQUS, the ABAQUS Interface for MSC.ADAMS reports the eigenvalues and frequencies of the modes it will store in the modal neutral file. As written to the screen during that translation step, the eigenvalues for the first eight modes are shown in Table 4–6.

Table 4–6 Eigenvalues computed by the ABAQUS Interface for MSC.ADAMS for the tire, using component modal synthesis with 20 vibration modes and 111 static modes.

Eigenvalue
-3741
-1969
0
0
0
0

Eigenvalue
3.139E+05
3.289E+05

The ABAQUS input files, `adams_ex3B.inp` and `adams_ex3C.inp`, are shown below.

`adams_ex3B.inp`

```

*heading
tire superelement w/ symmetric results transfer
step 0: generate full 3d model using tiretransfer_axi_full
step 1: equilibrate results
step 2: footprint analysis (displacement control)
step 3: footprint analysis (load control)
units: kg, m
*preprint,model=yes,history=yes
*node,nset=road
9999, 0.0, 0.0, -0.02
*symmetric model generation,revolve,element=200,node=200
0.0, 0.0, 0.0, 0.0, 1.0, 0.0
0.0, 0.0, 1.0
90.0, 3
70.0, 3
15.0, 7
10.0, 4
15.0, 7
70.0, 3
90.0, 3
*symmetric results transfer,step=1,inc=4
*elset,elset=foot,gen
1001, 4801, 200
1002, 4802, 200
1003, 4803, 200
1004, 4804, 200
1005, 4805, 200
1007, 4807, 200
1008, 4808, 200
1009, 4809, 200
1010, 4810, 200
1011, 4811, 200
1012, 4812, 200

```

EXAMPLES

```
1014, 4814, 200
*surface,type=cylinder,name=sroad
0., 0.,-0.31657, 1., 0.,-0.31657
0., 1.,-0.31657
start, -0.3, 0.
line, 0.3, 0.
*rigid body,ref node=9999,analytical surface=sroad
*surface,name=stread
foot, s3
*contact pair,interaction=srigid
stread, sroad
*surface interaction,name=srigid
*friction
0.0
*elset,elset=sect,generate
2800, 3200, 1
*nset,nset=sect,generate
2800, 3400, 1
*nset,nset=foot,elset=foot
*nset,nset=noutp,generate
1055, 5055, 200
*file format,zero increment
*****
*step,inc=100,nlgeom=yes
1: inflation
*static, long term
** 0.25, 1.0
1.,1.,1.
*model change, activate
*restart,write,overlay
*boundary
rim_ref, 1, 6
*dload
belt,p5, 200.e3
side,p5, 200.e3
*node print,nset=road,freq=100
u,
rf,
*el print,freq=0
*node file,nset=foot,freq=100
*output,field,freq=100
```

```

*element output
  s,le
*node output,nset=foot
  u,
*contact output, var=preselect
*output,history,freq=1
*node output, nset=road
  u, rf
*end step
*****
*step,inc=100,nlgeom=yes
  2: footprint (displacement controlled)
*static, long term
  0.2, 1.0
*restart,write,overlay
*print,contact=yes
*boundary,op=new
  rim_ref, 1, 6
  road, 1, 2
  road, 4, 6
  road, 3, , 0.02
*node print,nset=road,freq=100
  u,
  rf,
*el print,freq=0
*end step
*****
*step,inc=100,nlgeom=yes
  3: footprint (load controlled)
*static, long term
  1.0, 1.0
*boundary,op=new
  rim_ref, 1, 6
  road, 1, 2
  road, 4, 6
*cload,op=new
  road, 3, 3300.
*contact print
cdisp,cstress
*end step
*****

```

EXAMPLES

adams_ex3C.inp

```
*heading
  tire superelement w/ symmetric results transfer
Restart to identify nodes in footprint
  step 4: remove contact constraints
  step 5: extract fixed interface modes
  step 6: generate superelement
  units: kg, m
*preprint,model=yes,history=yes
*restart,read,step=3,write,overlay
*elset,elset=eall
tread,side,belt
**
*****
*nset,nset=footpr,unsorted
**
** This is the list of tire nodes found to be in contact with the
** road at the end of the previous step.
** (These nodes had status CL in the contact print table.)
**
  1850, 1855, 1905, 2045, 2050, 2055, 2100, 2105, 2245, 2250, 2255,
  2300, 2305, 2440, 2445, 2450, 2455, 2495, 2500, 2505, 2640, 2645,
  2650, 2655, 2695, 2700, 2705, 2840, 2845, 2850, 2855, 2895, 2900,
  2905, 3040, 3045, 3050, 3055, 3095, 3100, 3105, 3240, 3245, 3250,
  3255, 3295, 3300, 3305, 3440, 3445, 3450, 3455, 3495, 3500, 3505,
  3640, 3645, 3650, 3655, 3695, 3700, 3705, 3845, 3850, 3855, 3900,
  3905, 4045, 4050, 4055, 4100, 4105, 4250, 4255, 4305
*****
*nset,nset=footpr_retnodes
**
** This is the list of nodes in the above footprint that will be
** retained in the substructure.
**
  1850, 1855, 1905, 2045, 2100, 2440, 2445, 2450, 2455, 2505, 2500,
  2495, 3040, 3045, 3050, 3055, 3105, 3100, 3095, 3640, 3645, 3650,
  3655, 3705, 3700, 3695, 4045, 4050, 4105, 4100, 4250, 4255, 4305,
  2050, 2105
**
*****
```

```

*step,inc=1,nlgeom
  4: remove contact constraints
*static
1.,1.
*boundary,fixed,op=new
  rim_ref,1,6
  footpr,1,3
  road,1,6
*model change, type=contact pair, remove
  stread, sroad
**
** Write displacements for all nodes to the results file.
** (Needed so the MNF contains deformed nodal coordinates)
*node file
U,
*end step
**
*****
*step
  5: extract fixed interface modes
*frequency, eigensolver=lanczos
20,
**
*boundary,op=new
  road, 1, 6
  rim_ref,1,6
  footpr,1,3
**
** Write element mass matrices to the results file.
*element matrix output, mass=yes, elset=eall
**
** Write eigenvectors to the results file.
*node file
U,
*end step
**
*****
*step
  6: generate superelement
*substructure generate, type=z101, overwrite,
  recovery matrix=yes, mass matrix=yes

```

EXAMPLES

```
**
*boundary,op=new
  road, 1, 6
*retained nodal dofs, sorted=no
rim_ref,1,6
footpr_retnodes,1,3
*retained eigenmodes, generate
1,20,1
*substructure matrix output, stiffness=yes, mass=yes,
  load=yes, recovery matrix=yes
*end step
**
*****
```