Using Nastran Structural Models in Abaqus

Version 1.0 - 3/15/2022

Written by: Umut AKALP
Validated by: David PALMER
Edited by: Arati DESAI

SIMULIA Abaqus 2021 GA

DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Executive Summary

It is not uncommon for analysts to build workflows incorporating multiple software packages. For structural simulations, the capabilities of Abaqus and Nastran can be combined through the use of matrices. In particular, the linear structural models from Nastran may be brought into Abaqus in matrix form for additional linear or non-linear analyses.

This document provides two workflows for translating the Nastran structural models to Abaqus.

Prerequisites and recommendations: A basic knowledge of linear algebra and structural finite element analysis. Knowledge of linear dynamics modeling and substructuring (superelement or reduction techniques) methods is essential.

Target audience: Analysts that are generating Abaqus substructures from Nastran matrix data in either OP2 (binary) or DMIG (text) form.

Table of contents

1. Generating an Abaqus Substructure from Nastran Matrix Data in an OP2 File

This section focuses on translating system matrices from a Nastran OP2 file to Abaqus files that can be used in analyses.

For Nastran-to-Abaqus workflows, generating a substructure from Nastran data has two advantages:

1. It can be used in linear or non-linear, Static or Dynamic analyses, and,
2. It can be instanced multiple times in a model.

1.1. Preparing the Nastran Data for Translation

Nastran generates OP2 files in several formats. To create an OP2 file that can be read by the Nastran translator, the Nastran input file should contain:

```
ASSIGN OUTPUT2='modeldata.op2', UNIT=12
ASSIGN OUTPUT2='matrixdata.op2', UNIT=30
EXTSEOUT ( EXTID=10,
        STIFFNESS, MASS, LOADS,
        ASMBULK,
        EXTBULK, DMIGOP2=30 )
```

Due to changes in binary file formatting in some recent versions of Nastran, users may require to use the following keyword option to switch back to the previous formatting, which is essential for the Nastran-to-Abaqus workflow:

```
PARAM,SEOP2CV,1
```

The Nastran job with the keyword options shown above creates two files:

- *modeldata.op2* : This file contains the nodal coordinate data of the DMIG matrix data
- *matrixdata.op2* : This file contains the DMIG matrix data.

1.2. Translating Nastran Matrix Data to Abaqus

Using the *modeldata.op2* and *matrixdata.op2* files, you should run the Abaqus fromNastran translator using the following command:

```
abaqus fromNastran -job job_OP22SIM -op2file1 matrixdata.op2 -op2file2
modeldata.op2 -op2target GENERIC -verbose 1
```

With the '-verbose 1' option, the translator prints a detailed summary of the data blocks in the OP2 files. It also prints summary for a data block that is translated or skipped. The summary of data blocks in the command window is similar to:

```
Open OP2: matrixdata.op2
Found datablock "GEOM1X" (skipped)
```

```
Found datablock "GEOM2X" (skipped)
Found datablock "GEOM4X" (skipped)
Found datablock "MATK" (stiffness matrix)
Found datablock "MATM" (mass matrix)
Found datablock "MATP" (rforce data)
Found datablock "MATV" (recovery matrix column headers)
Found datablock "TUG1" (recovery matrix row headers)
Found datablock "MUG1" (recovery matrix data)
Found datablock "MUG1O" (recovery matrix data)
Found datablock "MUG1B" (recovery matrix data)
Found datablock "MUG1OB" (recovery matrix data)
 Open OP2: modeldata.op2
Found datablock "PVT" (skipped)
Found datablock "GPL" (FE mesh node numbers)
Found datablock "GPDT" (skipped)
Found datablock "EPT" (element properties)
Found datablock "MPT" (material properties)
Found datablock "GEOM2" (FE mesh element definition)
Found datablock "GEOM3" (load case definition)
Found datablock "GEOM4" (constraint definition)
Found datablock "GEOM1" (FE mesh node coordinates)
Found datablock "BGPDT" (FE mesh node coordinates)
Found datablock "DYNAMIC" (eigenvalues)
Found datablock "CASECC" (case control section)
Found datablock "LAMA" (eigenvalues)
Found datablock "BOPHIG" (skipped)
```

For the above example command, a successful translation generates a SIM file, namely job_OP22SIM.sim. This file contains the matrix information but it cannot be used as a substructure SIM file.

1.3. Generating an Abaqus Substructure from the Translated Matrix Data

The next step is to generate a full substructure library from the matrix data in the Abaqus SIM file. The translator does not generate a template input file for OP2 files, therefore you need to write a small input file for substructure generation. In some cases (see next workflow), the Abaqus fromNastran translator provides you with a template input file for substructure generation.

A short input file is needed, similar to the following:

```
*HEADING
*NODE, NSET=ALL_RETAINED_NODES
 1,      0.,      0.,       0.
    …
```

```
*MATRIX ASSEMBLE, STIFFNESS=KAAX, MASS=MAAX
*MATRIX INPUT, NAME=KAAX, MATRIX=STIFFNESS, INTERNAL=MODAL,
INPUT= job_OP22SIM.sim
*MATRIX INPUT, NAME=MAAX, MATRIX=MASS, INTERNAL=MODAL, INPUT=
job_OP22SIM.sim
*NSET, NSET=RETAINED_NODES_3_DOF
 1,2,…
** --------------------------------------------------
*STEP
*FREQUENCY
 3,
*BOUNDARY
RETAINED_NODES_3_DOF, 1,3, 0.
*END STEP
** --------------------------------------------------
*STEP
*SUBSTRUCTURE GENERATE, OVERWRITE, TYPE=Z1,
LIBRARY=op2_2_sim_SUBGEN, RECOVERY MATRIX=NO,
                    MASS MATRIX=YES
*RETAINED NODAL DOFS, SORTED=YES
 RETAINED_NODES_3_DOF, 1,3
*SELECT EIGENMODES, GENERATE
 1, 3, 1
*END STEP
```

1.4. Translation of the Modal Information from Nastran Files

The Abaqus fromNastran translator automatically appends the modal information to SIM files. Using '-verbose 1' option, the translator prints a summary similar to:

```
### SMASubNasToSim::c-tor: test.sim, Generic system
  Read STIFFNESS        : sparse, nonZeros 81
      matrix "STIF" has 3 SPOINTs
  Read MASS             : sparse, nonZeros 117
      matrix "MASS" has 3 SPOINTs
```

The three SPOINTS above correspond to the three eigenmodes stored in the Nastran OP2 file. Therefore, no further action is required.

2. Generating an Abaqus Substructure from Nastran DMIG Data in a Text (BDF) File

In this section, it is assumed that you already have a Nastran analysis that runs successfully and includes the EXTSEOUT keyword with the DMIGOP2 option to generate matrix data.

2.1. Formatting the Nastran Punch File for Translation

Nastran writes the substructure stiffness (and mass for a dynamic substructure) to a punch file starting with:

```
BEGIN SUPER     element_number
```

For the Abaqus fromNastran translator to read this file, place the following keyword options at the top of the punch file:

```
K2GG = KAAX
M2GG = MAAX
BEGIN BULK
```

To close the bulk data, add the following line at the end of the file:

```
ENDDATA
```

Save the modified file with .bdf extension and the substructure data is ready for the Abaqus fromNastran translator to read.

2.2. Running from Nastran Translator Using a Bulk Data File

Execute the translator using the following command:

```
abaqus fromNastran -job job_name -input bulk_data.bdf   -dmig2sim generic
```

Where, bulk_data.bdf is the substructure bulk data file from Nastran. Successful translation summarizes the written data in the command window if it is run interactively:

```
------------------------------------------------
DMIG data will be written to SIM as GenericSystem matrices (*MATRIX INPUT)
*** wrote       4 NODEs
NOTE: number of SPOINTs = 3
... writing *MATRIX INPUT for DMIG selected by K2GG=KAAX
... writing *MATRIX INPUT for DMIG selected by M2GG=MAAX
... writing STIFFNESS operator to SIM file job_name_X1.sim
... writing MASS      operator to SIM file job_name_X1.sim
found 81 DMIG real stiffness terms
found 117 DMIG mass terms
creating Abaqus input file job_name_X1_SUBGEN.inp
  (Run this input file to create a substructure from matrix data on job_name_X1.sim)
*** no elements in the model can have mass
WARNING: did not find "SOL=" in Nastran file
------------------------------------------------
```

The translator creates three files:

- **job_name.inp**: This is an input file that can be run in Abaqus if the Nastran bulk data file contains an analysis card (SOL 101 etc.). In this particular case, the bulk_data.bdf file only contains the substructure matrices. Therefore, this input file has a dummy STATIC step.
- **job_name_X1.sim**: This SIM file contains the substructure matrices in the MATRIX INPUT format. That is, these matrices can be used in an Abaqus analysis with the following option:

  *MATRIX INPUT, INPUT= job_name_X1.sim

  Note: the substructure matrices in the job_name_X1.sim file are not recognized as a substructure by Abaqus. A substructure can have multiple instances in an analysis, while the matrices in the above SIM file can only be used once in an analysis.

- **job_name_X1_SUBGEN.inp**: The translator also creates a substructure generation input file for convenience. This job uses the job_name_X1.sim file and generates a substructure library that can be used in Abaqus. Run this job as:

  abaqus -job job_name_X1_SUBGEN -interactive

For Abaqus 2022 and later versions, the substructure library consists of a single file:

1. job_name_Z1.sim

On the other hand, the substructure library consists of two files for Abaqus 2021 and earlier versions:

1. job_name_Z1.sim
2. job_name.sup.

2.3. Treatment of Internal Degrees of Freedoms and Modal Data

The retained Eigenmodes (QSETs in Nastran) are taken care of automatically by the fromNastran translator.

For example: for a Nastran job, the internal DOFs in the stiffness matrix are:

```
DMIG*   KAAX              900001          0
*          900001         0 5.764623106E+08
DMIG*   KAAX              900002          0
*          900002         0 8.011296053E+08
DMIG*   KAAX              900003          0
*          900003         0 2.614177711E+09
```

You can change the step definition in the translator output “job_name.inp” (see outputs of section 2.2 “Running fromNastran Translator Using a Bulk Data File”) as follows:

```
*STEP,PERTURBATION
*MATRIX GENERATE, STIFFNESS, MASS
*MATRIX OUTPUT,STIFFNESS, MASS
*END STEP
```

The step definition above prints the stiffness and mass matrices to the job_name_STIF1.mtx and job_name_MASS1.mtx files, respectively. The internal stiffness terms from Nastran are included as internal DOFs in the stiffness matrix. The corresponding internal DOFs in Abaqus will look like:

```
-3,1, -3,1,  5.764623106000000e+08
-2,1, -2,1,  8.011296053000000e+08
-1,1, -1,1,  2.614177711000000e+09
```

3. References

- NX Nastran User's Guide at https://docs.plm.automation.siemens.com/data_services/resources/nxnastran/10/help/en_US/tdocExt/pdf/User.pdf

- MSC Nastran User's Guide at https://simcompanion.mscsoftware.com/infocenter/index?page=content&cat=MSC__MD_Nastran_DOCUMENTATION&channel=DOCUMENTATION

- SIMULA User Assistance 2021

4. Document History

Document Revision	Date	Revised By	Changes/Notes
1.0	03/15/2021	Umut AKALP	Original