Translating Abaqus data to modal neutral file format for analysis in MSC.ADAMS

This example shows how you can investigate the flexibility of a substructure in your model during a dynamic analysis.

The following **abaqus adams** translator features are demonstrated:

- creating Abaqus models of MSC.ADAMS components,
- converting the Abaqus results into an MSC.ADAMS modal neutral (`.mnf`) file, the format required by ADAMS/Flex, and
- displaying results of an ADAMS/Flex flexibility analysis.

This page discusses:

Products: Abaqus/Standard

Application description

The following cases are illustrated:

- A three-dimensional steel link substructure meshed with solid elements and with displacement degrees of freedom at its nodes, as shown in Figure 1. Multi-point constraints connect it to the other components in the model. The analysis includes two steps: an eigenfrequency extraction step and a substructure generation step defined for a flexible body.
- The same steel link meshed with beam elements. Because the beam elements have both displacement and rotational degrees of freedom at their nodes, no multi-point constraints are required to connect the substructure to the rest of the model. The analysis includes two steps: an eigenfrequency extraction step and a substructure generation step defined for a flexible body.

Abaqus modeling approaches and simulation techniques

Both cases described in this section share the same general approach:

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 adams** translator to read the Abaqus results from the SIM database 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.

Summary of analysis cases

Case 1	Link modeled with solid elements.
Case 2	Link modeled with beam elements.

Case 1 Link modeled with solid elements

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

Analysis types

The example includes an eigenfrequency extraction and a substructure generation analysis.

Mesh design

The link is modeled with 642 C3D10 tetrahedral solid elements (1368 nodes).

Materials

The steel used in this case has a Young's modulus of 2.07×10^{11} N/m^2 (3.0×10^7 lbf/in^2) and a Poisson's ratio of 0.29. The density of the model is 7.8×10^3.

Boundary conditions

The `RETNODES` node set is fixed in the 1- and 6-directions.

Constraints

This analysis includes two multi-point constraints: one applied to the `LEFTCYL` node set and the other applied to the `RIGHTCYL` node set.

Analysis steps

The analysis includes two steps: an eigenfrequency extraction step and a substructure generation analysis step.

Output requests

Element stiffness matrices and mass matrices are written to the SIM database for the element set `PROP1` as part of the substructure generation analysis step.

Run procedure

You can perform the analysis of the link with solid elements using the procedure shown below.

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 adams** translator and translate the results in a SIM database generated in the Abaqus analysis to a modal neutral file for use with ADAMS/Flex:

```
abaqus adams job=adams_ex1 substructure_sim=adams_ex1_Z1
```

Results and discussion

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.

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 1. These frequencies are reported in the `adams_ex1.dat` file. The **abaqus adams** translator 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 2. These frequencies are written to the screen when executing the **abaqus adams** translator.

Case 2 Link modeled with beam elements

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

Analysis types

As in Case 1, this example includes an eigenfrequency extraction and a substructure generation analysis.

Mesh design

The mesh for the beam model uses 10 B31 elements (11 nodes).

Materials

The steel material definition is the same as in Case 1.

Boundary conditions

The beam elements have both displacement and rotational degrees of freedom at their nodes.

Analysis steps

The analysis includes two steps: an eigenfrequency extraction step and a substructure generation analysis step.

Run procedure

You can perform the analysis of the link with beam elements using the procedure shown below.

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 adams** translator and translate the results in a SIM database generated in the Abaqus analysis to a modal neutral file for use with ADAMS/Flex:
   ```
   abaqus adams job=adams_ex2 substructure_sim=adams_ex2_Z1
   ```

Results and discussion

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.

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

Input files

Case 1 Link modeled with solid elements

adams_ex1.inp

Input file to analyze a link model subjected to a gravity load.

adams_ex1_nodes.inp

Node definitions for Case 1.

adams_ex1_elements.inp

Element definitions for Case 1.

Case 2 Link modeled with beam elements

adams_ex2.inp

Input file to analyze a link model subjected to a gravity load.

Tables

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

Frequency, Hz
206
391
570
1124
1228
1817
1879
2541

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

Frequency, Hz
194
535
574
1055
1551

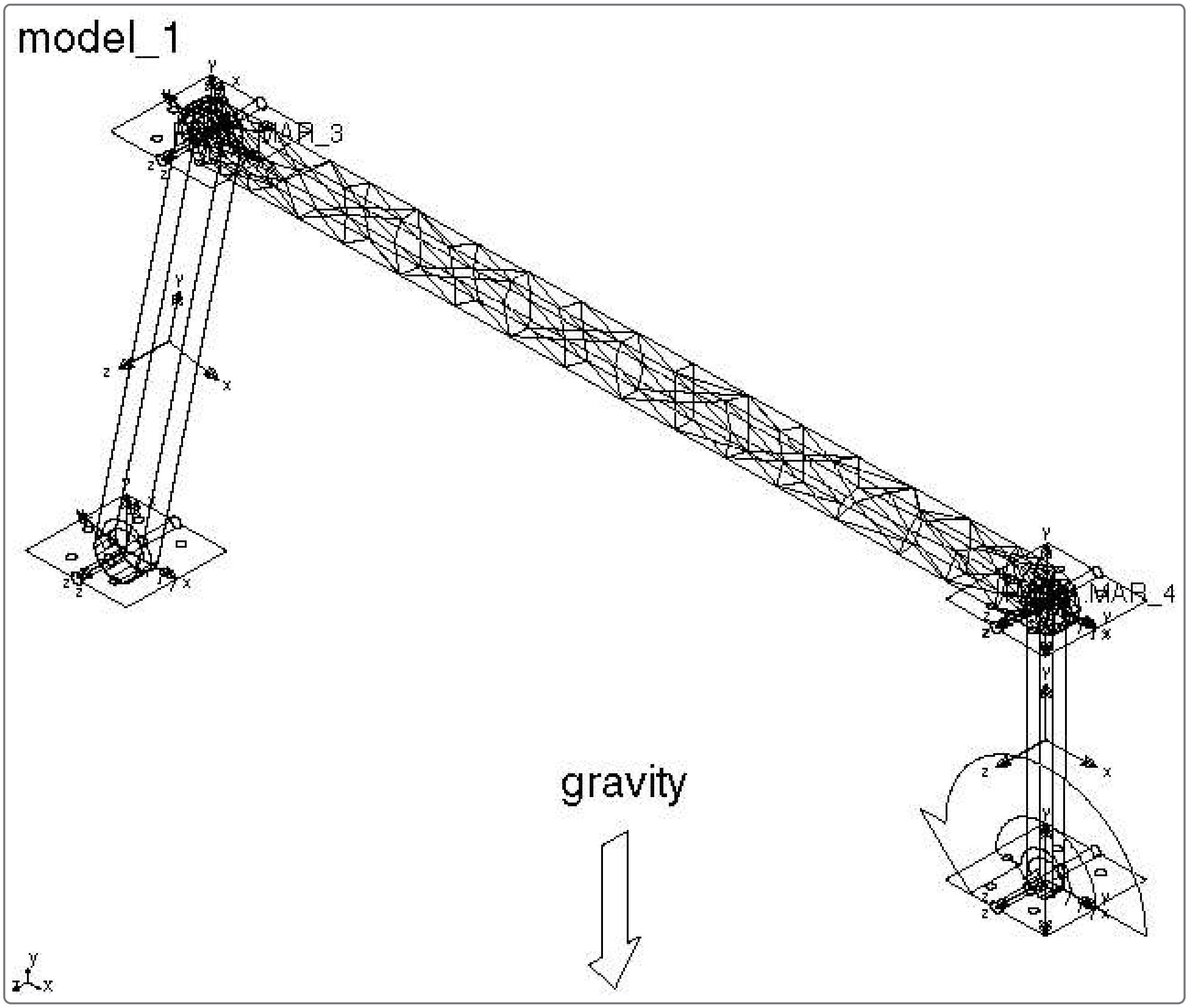
Frequency, Hz
1762
1801
2653

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

Frequency, Hz
205
555
610
1070
1618
1742
1775
2568

Figures

Figure 1. Solid link model.


model_1
MAR_3
MAR_4
gravity