Generating Flexible Body Standard Input Data from an Abaqus Substructure

Standard Input Data (SID) describes models for simulation of flexible members in multi-body analysis codes, such as DYMOLA. The SID is based on the modal representation of small deformations and can be generated from an Abaqus substructure in Releases 6.10 and 6.10-EF.

Keyword options to generate SID

To enable SID generation, a substructure has to be generated with an undocumented option:

```
*SUBSTRUCTURE GENERATE, FLEXIBLE BODY=S
```

After this the SID generation tool can be called to generate SID from a substructure SIM file.

Command line interface for the SID generation tool

Usage:

```
abq tosid -job <jobName> [optional parameters]
```

Required arguments:

```
-job
```
         : jobName is name of the existing Abaqus substructure

Optional arguments:

`-slength`	: scaling factor for the length unit (default: 1.0)
`-smass`	: scaling factor for the mass unit (default: 1.0)
`-stime`	: scaling factor for the time unit (default: 1.0)
`-fmin`	: lower boundary of the frequency range (default: 0.001 Hz)
`-fmax`	: higher boundary of the frequency range (default: 1.E16 Hz)
`-tol`	: zero cutoff tolerance (default: 1E-12)
`-help`	: this usage info

The **job** parameter specifies the name of Abaqus substructure. This name is the substructure SIM name – i.e., the name of the substructure `.sim` file. By Abaqus conventions the substructure name includes "_Zxxx," where xxx is a 1-to-3 digit number. For example, the substructure `.sim` file may have the name `Qertraeger1_Z1.sim`.

In this case the command must be

```
abq toSid -job Qertraeger1_Z1
```

The **slength**, **smass**, and **stime** parameters allow unit conversion to be done during SID generation.

Parameters **fmin** and **fmax** specify the frequency range for the substructure mode selection at SID generation. Parameter **tol** specifies zero cutoff tolerance. All entries with absolute values below this tolerance will be ignored in the generated SID entities.

Additional controls are available for the SID generation process. These controls can be optionally provided in the text file *substructureName*.inp in the form of Abaqus keyword options. For example, in the case of substructure `Qertraeger1_Z1.sim` the input file containing the SID generation controls must be called `Qertraeger1_Z1.inp`.

Usage of the keyword options allowed for SID generation follows the conventional documented rules, but only a limited number of keywords can be used, and a limited number of features provided by these keywords are processed. The data for the keyword options references the generated substructure. Thus, node labels are the substructure retained node labels, mode numbers are the substructure eigenmode numbers (for eigenmodes extracted with the condensed substructure stiffness and mass), etc. Keyword options supported for the SID generation process are listed below with all possible parameters and data options.

*NSET

Option ***NSET** can be used to assign the substructure retained nodes to a node set.

Required parameters:

NSET

Set this parameter equal to the name of the node set to which the nodes will be assigned.

Node sets can be used for convenience in other keyword options. A node set with the name "SID_SELECTED_NODES" plays a special role in the SID generation. If a node set with the name "SID_SELECTED_NODES" is specified, then the retained node data will be written selectively only for the nodes from this node set in the "frame" section of the SID file. If the node set "SID_SELECTED_NODES" is not specified then the retained node data is generated for all retained nodes of a substructure.

Optional parameters supported for SID generation:

GENERATE

If this parameter is included, each data line should give a first node, *n1*; a last node, *n2*; and the increment in node numbers between these nodes, *i*. Then, all nodes going from *n1* to *n2* in steps of *i* will be added to the set. *i* must be an integer such that *(n2-n1)/i* is a whole number (not a fraction).

Data lines if the GENERATE parameter is omitted:

First line:

1. List of nodes or node set labels to be assigned to this node set. Only previously defined node sets can be assigned to another node set. Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if the GENERATE parameter is included:

First line:

1. First node in the set.
2. Last node in the set.
3. Increment in node numbers between nodes in the set. The default is 1.

Repeat this data line as often as necessary.

*BOUNDARY

Option ***BOUNDARY** can be used to prescribe zero boundary conditions at the substructure retained nodes for the substructure eigenmodes extraction during SID generation. If this option is not used, then the eigenmodes of the unsupported substructure are extracted.

No parameters are used for SID generation.

Data lines to prescribe boundary conditions (only the "direct" format is supported):

First line:

1. Node number or node set label.
2. First degree of freedom constrained.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

Repeat this data line as often as necessary to specify boundary conditions at different substructure retained nodes and degrees of freedom.

*SELECT EIGENMODES

Option ***SELECT EIGENMODES** selects the substructure eigenmodes to be used for the flexible body representation. If this option is not used, then all the substructure eigenmodes are used in the flexible body SID generation.

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the selected eigenmodes are given as a collection of mode numbers.

Set DEFINITION= FREQUENCY RANGE to indicate that the modes are selected from the specified frequency ranges including frequency boundaries. Frequency ranges can be discontinuous.

GENERATE

If this parameter is included, each data line should give a first mode, *n1*; a last mode, *n2*; and the increment in mode numbers between these modes, *i*. Then, all modes going from *n1* to *n2* in steps of *i* will be added to the set. *i* must be an integer such that *(n2-n1)/i* is a whole number (not a fraction).

This parameter can be used only with DEFINITION=MODE NUMBERS.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is omitted:

First line:

1. List of modes to be used.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is included:

First line:

1. First mode number.
2. Last mode number.
3. Increment in mode numbers between modes. The default is 1.

Repeat this data line as often as necessary.

Data lines if DEFINITION=FREQUENCY RANGE:

First line:

1. Lower boundary of the frequency range (in cycles/time).
2. Upper boundary of the frequency range (in cycles/time).

Repeat this data line as often as necessary.

*DAMPING CONTROLS

Option ***DAMPING CONTROLS** can be used to control the type of the substructure damping used for the flexible body SID generation.

Optional parameters supported for SID generation:

VISCOUS

Set this parameter equal to ELEMENT to request the condensed substructure viscous damping matrix.

Set this parameter equal to FACTOR to request a viscous damping matrix that includes the substructure mass and stiffness proportional Rayleigh damping with the factors ALPHA and BETA specified under the *DAMPING option.

Set this parameter equal to COMBINED to request a viscous damping matrix that includes a combination of ELEMENT and FACTOR.

Set this parameter equal to NONE to exclude the viscous damping matrix completely.

If this parameter is omitted or the option is not used, the COMBINED damping is used.

*DAMPING

Option ***DAMPING** can be used in conjunction with the ***DAMPING CONTROLS** option to specify the factors ALPHA and BETA, the substructure mass and stiffness proportional Rayleigh damping.

Optional parameters supported for SID generation:

ALPHA

Set this parameter equal to the factor to create Rayleigh mass proportional damping. The default is ALPHA=0.

BETA

Set this parameter equal to the factor to create to create Rayleigh stiffness proportional damping. The default is BETA=0.