Translating an Abaqus substructure to a Simpack flexible body

The **abaqus tosimpack** translator converts an Abaqus substructure to a flexible body in a format that can be used by the Simpack multibody dynamics solver.

The translator reads Abaqus data from a substructure SIM file and writes data to a Simpack Flexible Body Interface (FBI) file.

The following topics are discussed:

- Using the translator
- Command summary
- Command line options

Related Topics

About Execution Procedures

Products: Abaqus/Standard

Using the translator

The following procedure summarizes the typical usage of the **abaqus tosimpack** translator:

1. Create an Abaqus substructure. (General guidelines for building Abaqus models with substructures are described in Using substructures.)

   The substructure generation step must write at least the mass and stiffness matrices. It can also write the recovery, viscous damping, and structural damping matrices. The *FLEXIBLE BODY option must be used. For example,

   ```
   *SUBSTRUCTURE GENERATE, MASS MATRIX=YES, RECOVERY MATRIX=YES, VISCOUS DAMPING MATRIX=YES,
   STRUCTURAL DAMPING MATRIX=YES
   *FLEXIBLE BODY, TYPE=SIMPACK
   ```

   In addition, you can add the following data to translate stress:

   ```
   *ELEMENT RECOVERY MATRIX, POSITION=AVERAGED AT NODES
   S,
   ```

   You can recover results on a subset of the model using the ELSET parameter for the substructure generation and element recovery matrix options shown above.

   The Simpack FBI file is always in SI units. If the Abaqus analysis does not use SI units (meters, kilograms, and seconds), you can define the model units using the *UNIT SYSTEM option. Alternatively, you can define the model units when you run the translator using the **length**, **mass**, and **time** command line options. If the Abaqus analysis uses American or English units (inches, pounds force, and seconds), the mass unit is slinch.

2. Run the Abaqus analysis.

3. Run the **abaqus tosimpack** translator to read the substructure SIM database produced by the analysis and to create the flexible body interface file.

▲

Command summary

abaqus tosimpack

> **job** = *job-name*
>
> [**substructure_sim** = *filename*]
>
> [**length** = *length-units-name*]
>
> [**mass** = *mass-units-name*]
>
> [**time** = *time-units-name*]

▲

Command line options

job

This option specifies the input and output file names to use during results translation. The *job-name* value is used to construct the default substructure SIM database file name, *job-name*.sim. The output flexible body interface file is given the name *job-name*.fbi.

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

substructure_sim

This option specifies the name of the substructure SIM database (`.sim`) file if it is different from `job-name`.sim.
The file will usually be named `job-name_Znn`.sim.

length

This option specifies the **length** units for the model. The valid options are as follows:

- `m, meter, meters`
- `mm, millimeter, millimeters`
- `cm, centimeter, centimeters`
- `km, kilometer, kilometers`
- `in, inch, inches`
- `ft, foot, feet`
- `mile, miles`

mass

This option specifies the **mass** units for the model. The valid options are as follows:

- `kg, kilogram, kilograms`
- `mgg, megagram, megagrams, tonne, tonnes`
- `gram, grams`
- `lbm, pound, pounds, pound_mass`
- `uston, ustons, us_ton`
- `klbm, kpound_mass`
- `ozm, ounce, ounces, ounce_mass`
- `slug`
- `slinch, dozen_slug`

time

This option specifies the **time** units for the model. The valid options are as follows:

- `s, sec, second, seconds`
- `ms, millisecond, milliseconds`
- `min, minute, minutes`
- `h, hr, hour, hours`