Dynamic Design Analysis Method (DDAM) Response Spectrum Analysis with Abaqus

Summary

The Dynamic Design Analysis Method (DDAM) is a U.S. Navy methodology for qualifying shipboard equipment and supporting structures for survival of shock loading due to underwater explosions (UNDEX). The DDAM is a regimented collection of procedures that utilize estimates of the peak linear dynamic response of ship equipment and structures obtained from normal mode response spectrum analyses. The response spectra are usually based upon empirically obtained Shock Design Values (SDVs) that are dependent on the type of ship, mounting location, direction of attack, response frequencies, and overall survivability requirements.

The requirements for efficiently implementing the DDAM procedures are quite extensive. However, the customization capabilities with Abaqus/CAE are ideally suited for streamlining the process. In this technology brief a custom GUI application, designated as DDAM for Abaqus, is presented. It is designed to capitalize on the excellent normal mode capabilities within Abaqus and allows engineers to perform DDAM analyses efficiently. DDAM for Abaqus is available with the release of Abaqus Version 6.5-3 and is free and unlicensed for existing Abaqus sites.

Background

The DDAM is used to evaluate the survivability of ship equipment and supporting structures when the dynamic response is predominantly linear. The DDAM is not applicable when the dynamic response is highly nonlinear, such as when equipment is mounted resiliently or extensive material plasticity is expected. The following tasks are associated with the application of the DDAM:

1. FEA model creation

2. Frequency extraction

3. Mode selection

4. Spectrum definition

5. Response calculation

6. Results evaluation

7. Report writing

8. Design review
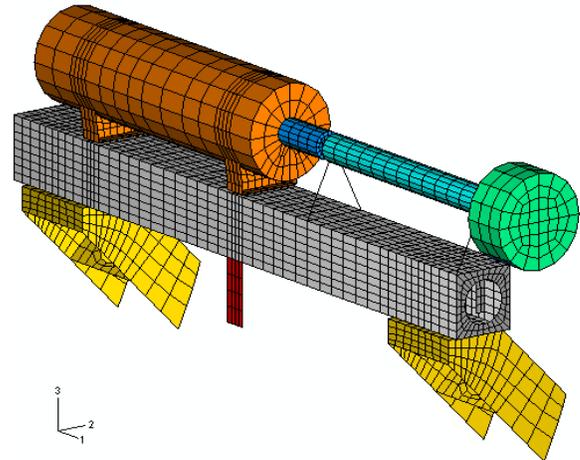
Abaqus model of a motor-pump assembly located on a hull-mounted box girder.

Key Abaqus Features and Benefits

- Provides a complete, convenient environment in which to perform DDAM analyses with Abaqus

- Assists with mode selection, spectrum definition, and response calculation

- Incorporates U.S. Navy-specific analysis procedures

- Features NRL, SRSS, and ABS modal summation methods

- Accommodates both DDAM and DDAM-CSM solution method options

- Provides an SSCC catalog consistent with NRL-1396, but also accommodates user-defined catalogs for computing SDVs

- Features both automatic and manual modal selection

- Provides output of individual modal responses and modal summed peak responses

- Automatically generates summary tables, modal weight plots, and response spectra plots

DDAM for Abaqus was developed to assist engineers in meeting DDAM requirements, primarily in the areas of mode selection, spectrum definition, and response calculation.

DDAM for Abaqus is a custom Abaqus/CAE module. The DDAM for Abaqus Graphical User Interface (GUI) is located in the Abaqus/CAE Visualization module. In addition to the aforementioned functionality, pertinent DDAM-related output is also generated to support the results evaluation and report writing tasks.

Analysis Approach

In a DDAM for Abaqus analysis, three primary tasks must be completed. Each has its own task manager.

1. Create DDAM spectrum

2. Create DDAM model

3. Create DDAM job (calculate response)

The motor-pump assembly model shown on Page 1 is used to illustrate how these primary tasks are completed within DDAM for Abaqus.

Create DDAM Spectrum

The DDAM response spectrum is an acceleration spectrum defined in terms of the gravitational constant, g. The data points that comprise the DDAM response spectrum are designated as SDVs and are a function of the following geometric, modal analysis, UNDEX loading, and survivability factors:

- Ship type (surface or submarine)

- Mounting location (hull, deck, or shell plating)

- Attack direction (vertical, athwartship, or fore-aft)

- Modal weights

- Deformation (elastic or elastic-plastic)

The **Create DDAM Spectrum** dialog box is shown in Figure 1. The DDAM for Abaqus procedure for computing the SDVs is the same as that detailed in Naval Research Laboratory Memorandum Report NRL-1396 (Ref. 2). The tables and reference equations presented in NRL-1396 are incorporated into two evaluation equations, one representing a velocity limit and the other an acceleration limit on the shock response. The nine coefficients needed to define these two equations are called the Shock Spectrum Calculation Constants (SSCC). The nine SSCC are selected from a catalog or specified via direct input. DDAM for Abaqus provides an SSCC catalog consistent with NRL-1396; however, user-defined catalogs can also be created.
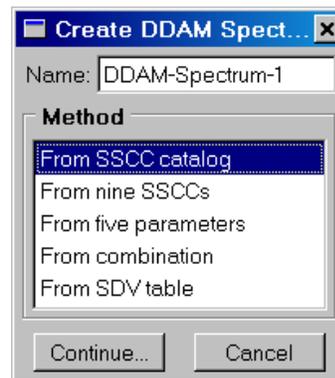

*Figure 1: **Create DDAM Spectrum** dialog box.*

For those cases where a response spectrum appropriate for a submarine is required, an option has been included to provide five independent parameters for computing the SDVs.

A response spectrum that is a combination of any previously created spectra can also be created. In this case DDAM for Abaqus calculates the peak response due to each individual spectrum in the combination and writes the extreme peak response over all of the spectra to the DDAM response output database. As a final option, the SDVs can be input directly.

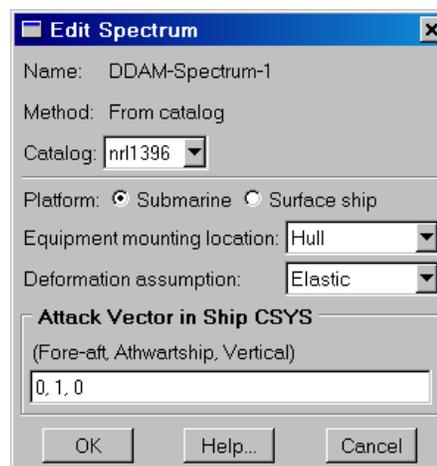Figure 2 shows the dialog box used to create a spectrum from the NRL-1396 SSCC catalog.

*Figure 2: **Edit Spectrum** dialog box.*

The **Edit Spectrum** dialog box includes information on the platform, equipment mounting location, deformation assumption, and the direction of the attack. This information is required irrespective of the option being used to create the spectrum, since it is used to ensure that the DDAM model created for analysis is consistent with the defined response spectrum. The direction of the attack does not have to be aligned with a ship axis. SDVs for oblique attack directions are derived as discussed in Appendix D of NAVSEA 0908-LP-000-3010 Rev. 1.

Create DDAM Model

DDAM for Abaqus performs response spectrum analyses by operating on the results of an Abaqus normal mode frequency extraction. In creating a model, DDAM for Abaqus accesses the Abaqus input file and frequency extraction output database (`.odb`) file that represent the equipment and supporting structures being evaluated. The following information completes the DDAM model:

- Model location/orientation within the ship
- DDAM solution controls
- Mode selection criteria
- DDAM response output request

These data are input to DDAM for Abaqus via a series of tabbed pages associated with the **Create DDAM Model** dialog box, as shown in Figure 3.
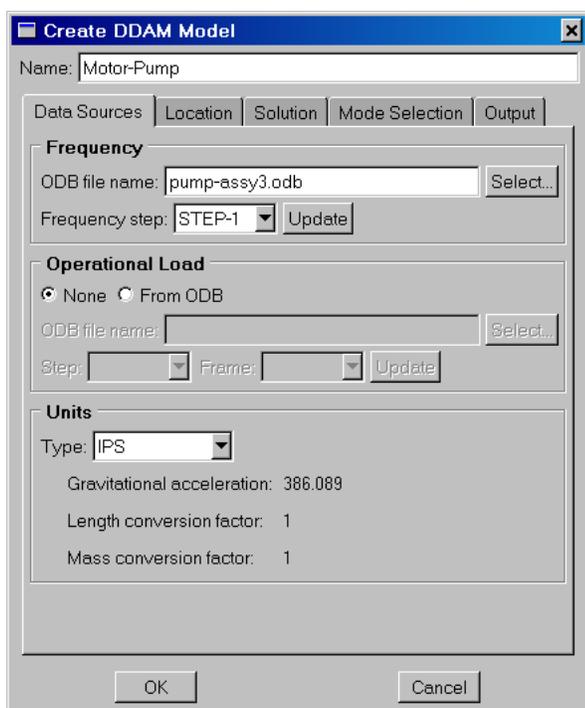

Figure 3: **Create DDAM Model** *dialog box.*

Create DDAM Model: Data Sources

The output database file that contains the normal modes frequency results is specified in the **Frequency** portion of the **Data Sources** tabbed page. The output database file can be scanned to identify which frequency extraction steps are available for the DDAM spectrum analysis.

If operational load stresses are to be combined with the DDAM results, the appropriate output database file, anal-ysis step, and frame that contains the operational stresses are specified in the **Operational Load** portion of the **Data Sources** tabbed page.

The system of units used with the Abaqus finite element model is specified in the **Units** portion of the **Data Sources** tabbed page. The units type can be IPS (Inch-Pound-Sec), FPS (Foot-Pound-Sec), MKS (Meter-Kilogram-Sec), CGS (Centimeter-Gram-Sec), or a set of user-defined units. DDAM for Abaqus converts the FEA model effective modal masses to modal weights with units of kips when determining the SDVs. However, the DDAM results are generated and presented with the FEA model units.

Create DDAM Model: Location

The location and orientation of the finite element model with respect to the ship is defined as shown in Figure 4.

The **Platform** and **Mounting location** defined on this tabbed page correspond to the spectrum to be used in the DDAM analysis. The orientation of the ship axes with respect to the finite element model global Cartesian axes can be defined in terms of direction cosines or vectors that represent the ship's fore-aft and athwartship directions.
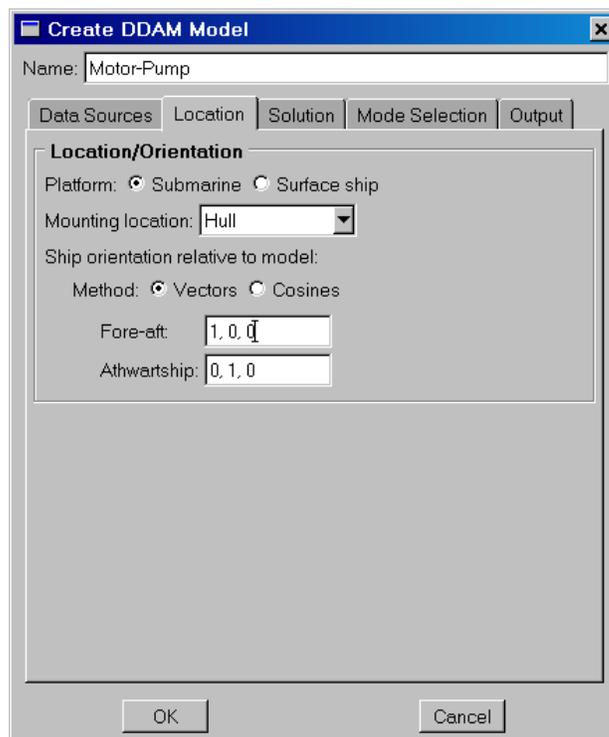

Figure 4: Specification of location and orientation of the finite element model.

Create DDAM Model: Solution

The solution options are selected on the **Solution** tabbed page, shown in Figure 5. The **Shock spectrum** to be used for the analysis is selected from a pull-down menu, and the minimum shock response g acceleration level is input (default = 6). Three modal **Summation method** options are available: NRL, SRSS, and ABS. The default is NRL, which represents the Naval Research Laboratory modal summation method, currently the recommended method for performing DDAM response spectrum analyses. SRSS is the classical Square Root of the Sum of the Squares method, and ABS is the classical Absolute Summation method.
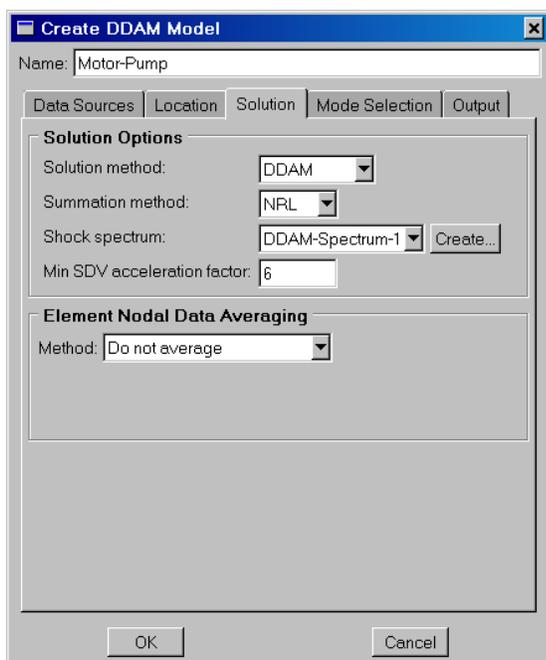
Figure 5: Specification of solution options.

Two **Solution method** options are available: DDAM and DDAM-CSM. Selecting DDAM indicates that the standard NRL summation approach will be utilized. DDAM-CSM (Closely Spaced Modes) is selected when performing an alternative modal summation response assessment that accounts for the behavior of closely spaced modes. The present motor-pump example uses the DDAM solution method.

The DDAM-CSM option processes any out-of-phase modal response with the Closed Form Treatment detailed in NAVSEA 0908-LP-000-3010. Mode pairs for this treatment are defined as having frequencies within 10% of their common mean frequency. The Closed Form Treatment is applied only to response quantities that are out-of-phase. When Mises stress is processed, the sign on the pressure stress is used to determine the modal phasing.

DDAM-CSM is also used to invoke an algebraic sum approach for processing classical repeated modes or closely spaced modes that are dominated by in-phase behavior. The DDAM-CSM algebraic sum approach can be applied only to mode pairs that have frequencies within 1% of their common mean frequency. The mode pairs that will be processed with the DDAM-CSM methodology are specified in the **Closely Spaced Modes** portion of the **Solution** tabbed page, as shown in Figure 6. Modes processed with the Closed Form Treatment are identified as **Out-of-phase mode pairs**, and those processed with the algebraic sum approach are identified as **Repeated mode pairs**. The **Critical damping fraction** used with the Closed Form Treatment is also specified.

Three choices are provided for **Element Nodal Data Averaging** when computing the peak DDAM response. The peak response quantities are stored in the DDAM response output database file as nodal values on an element-by-element basis. The **Do not average** option indicates that the nodal data stored for each element are unique and are not influenced by the adjacent elements. The **Average all** option indicates that the nodal values stored are the result of averaging values from all the adjacent elements. The **Average only within regions** option indicates that the nodal values stored are the result of averaging values from a selected group of adjacent elements. You can define the groups of elements used in this averaging scheme; otherwise, default element sets that are based on the element section property definitions are used.
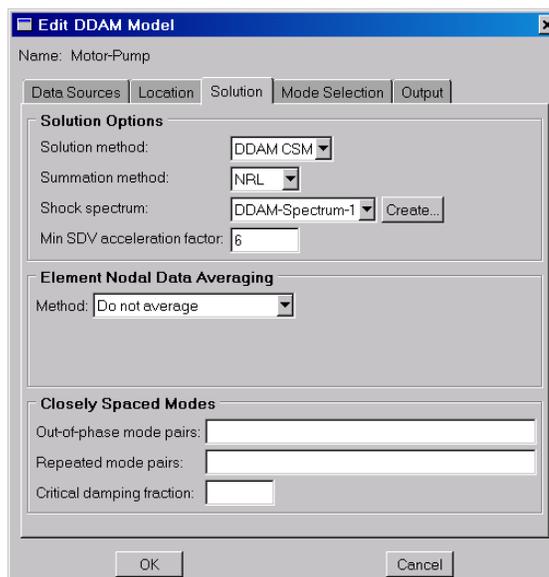

Figure 6: DDAM-CSM solution options.

Create DDAM Model: Mode Selection

The **Mode Selection** tabbed page shown in Figure 7 contains **Automatic** and **Manual** portions. The **Automatic Selection Criteria** can be set to either NAVSEA 3010, which invokes the procedures of NAVSEA 0908-LP-000-3010 Rev. 1, or **NONE**, in which case only the **Manual** portion remains active.
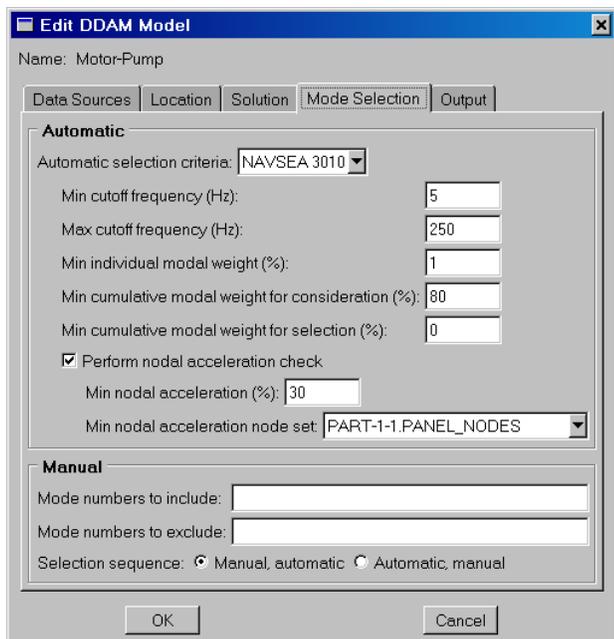

Figure 7: Mode selection options.

The NAVSEA 3010 Automatic Mode Selection procedure requires listing a minimum and maximum cutoff frequency for the modes to be considered for inclusion in the response spectrum analysis. The default values are 5 Hz and 250 Hz. There is also a requirement that the cumulative modal weight of all modes within the cutoff frequency range must be greater than a specified value, with the NAVSEA 3010 default being 80%. This cumulative modal weight criterion is for identifying the modes that will be considered for selection and is given the label **Min. cumulative modal weight for consideration**. However, the principal criterion for selecting modes, designated as the **Min. individual modal weight**, selects modes in which the effective modal weight is greater than some minimum percent of the total system weight, with the NAVSEA 3010 default being 1%.

In some cases design contracts will include an additional requirement that enough modes are selected such that the cumulative modal weight of all the selected modes exceeds a percentage of the total system weight. This is not a strict NAVSEA 3010 requirement and is essentially turned off by the default setting of 0% for the parameter **Min. cumulative modal weight for selection**.

Once the modes to be included in the response spectrum analysis have been determined based on the modal weight and cut-off frequency requirements, NAVSEA 3010 provides for the selection of additional modes based on a nodal acceleration check. The acceleration check is designed to account for the localized response of relatively lightweight subcomponents that may represent critical areas of concern for shock design.

The additional modes to be included under the acceleration check are those in which a nodal acceleration exceeds by some amount (NAVSEA 3010 default = 10%) the maximum nodal acceleration from any of the modes selected via the modal weight criteria. Only the acceleration response of nodes in the user-identified critical areas need to be considered. For the motor-pump example the acceleration check is active with a 30% criteria and a critical region of nodes for the small cantilevered panel directly beneath the motor (labeled **PART-1-1.PANEL_NODES** in the **Automatic** portion of the **Mode Selection** tabbed page).

You can manually select modes to supplement the modes selected automatically via NAVSEA 3010. The mode numbers from the frequency extraction to be included or excluded from the response spectrum analysis are specified in the **Manual** portion of the **Mode Selection** tabbed page. The **Selection Sequence** option allows the specification of the order in which the **Automatic** and **Manual** selection criteria are applied. The **Manual, automatic** option adds the included modes to the selected set and removes the excluded modes from consideration for the selected set prior to the application of the modal weight and acceleration check criteria. The **Automatic**, **manual** option includes and excludes the listed modes after the automatic selection process has been completed.

Create DDAM Model: Output

DDAM for Abaqus accesses the results of a frequency extraction analysis from the output database file, which as a minimum must contain nodal displacements for all the modes that may need to be included in the DDAM response spectrum analysis. In addition, the mode shapes obtained from the frequency extraction analysis must have been normalized with respect to the finite element model's mass matrix.

DDAM for Abaqus can use the mode shape displacement data within the frequency output database to compute the modal summed peak response for displacements, velocities, and accelerations. DDAM for Abaqus also supports the computation of modal summed peak response for indirect solution variables such as stresses, strains, section forces, and moments.

DDAM for Abaqus initially tries to obtain the modal results

data needed for processing indirect solution variables from the frequency extraction output database. If the required data are not stored in this database, DDAM for Abaqus submits an auxiliary Abaqus/Standard job to extract the data only for those modes that will be used in the DDAM response spectrum summation.

DDAM for Abaqus requires the specification of the output variables for which modal summed peak responses will be computed. Figure 8 shows the **Output** tabbed page used to create and summarize output requests.
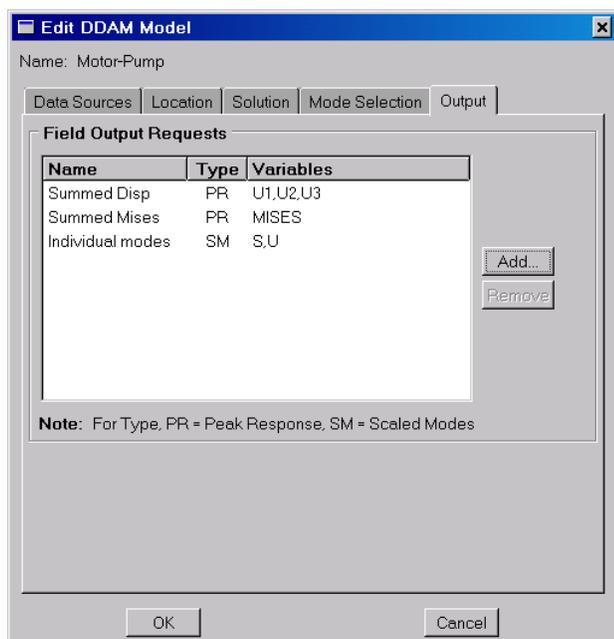

Figure 8: Output request specification.

All output quantities that are to be summed over the selected modes are treated as scalars during the summation process. Therefore, derived quantities such as Mises stress are computed on a mode-by-mode basis, with the summation processes using the modal values, as required by NAVSEA. This is different from the Abaqus/Standard response spectrum solver procedure, where the peak Mises value is derived from the modal summed stress components.

Figure 8 also shows that there are two types of output requests associated with DDAM for Abaqus. The designation **PR** indicates that the modal summed peak response will be stored in Step 1 of the DDAM response spectrum output database. The **SM** designation indicates that the requested output (scalar, vector, or tensor) will be generated for a set of user-specified modes, with the values scaled to represent the peak response of each mode. These data are stored in Step 2 of the DDAM response spectrum output database, with a separate frame of data for each of the user-specified modes.

The **Add** button is used to open the **Add Output Request** dialog box. Figure 9 shows this dialog box as used to request the Mises peak response (**PR**) as DDAM output. Figure 10 shows this dialog box as used to create the scaled mode (**SM**) request for the motor-pump example. As shown in Figure 10, you can make a scaled mode output request based on all of the modes previously selected or you can specify a different set.
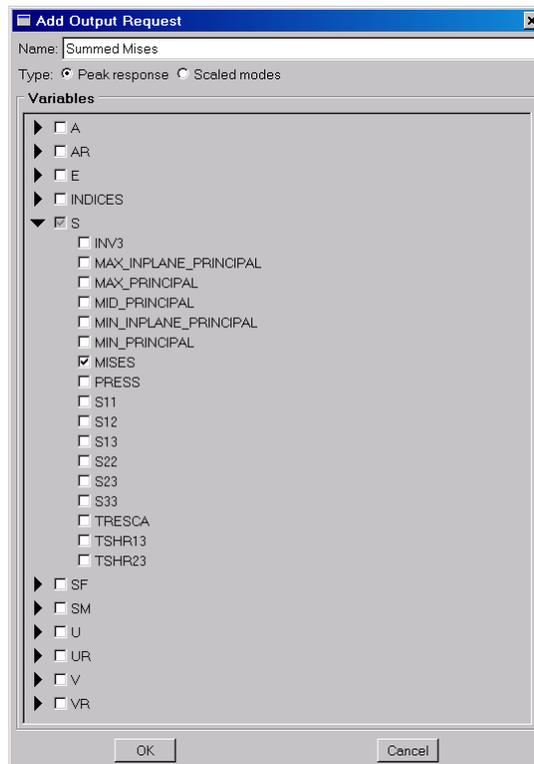

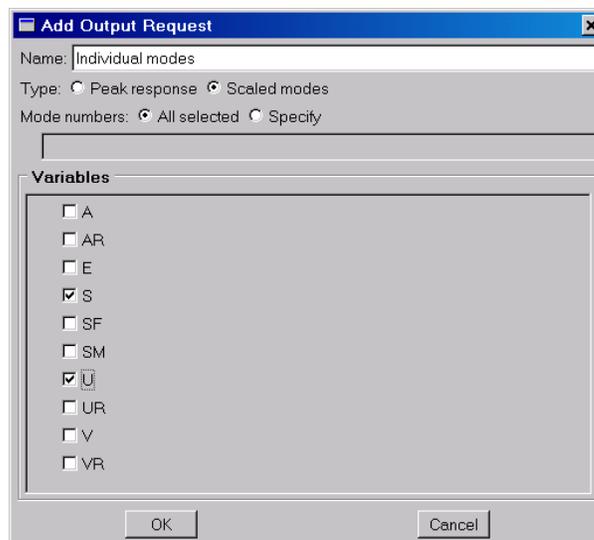*Figure 9: Peak response output request.*

Figure 10: Scaled modes output request.

The **INDICES** output variable is specific to DDAM for Abaqus and is used to indicate which of the selected modes contributes the most to the peak response. When this output option is requested, a mode index is generated for each of the requested peak response output variables. The index number corresponds to indices that are assigned to the selected modes as listed in the DDAM for Abaqus-generated `Table4.html` output file.

Clicking **OK** in the **Edit DDAM Model** dialog box completes the creation of the DDAM model. More than one DDAM model can be created and stored within an Abaqus/CAE database, and the currently defined models can be listed with the **DDAM Model Manager**, as shown in Figure 11.
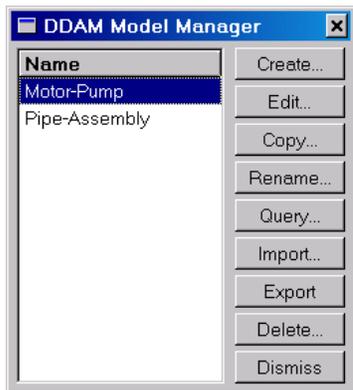
*Figure 11: **DDAM Model Manager**.*

The **Export** and **Import** options allow the DDAM modal data to be written to and read from an ASCII archive file. The **Query** option opens the **Query DDAM Model** dialog box (Figure 12), which is used to obtain graphical and tabular output that summarizes the results of the frequency extraction, mode selection process, generated SDVs, and modal weights. These data are created prior to the initiation of the actual DDAM response spectrum analysis, where the modal summation for predicting the peak response actually takes place. The **Plot Mode** option produces a displaced shape plot of the indicated mode with four standard viewports.
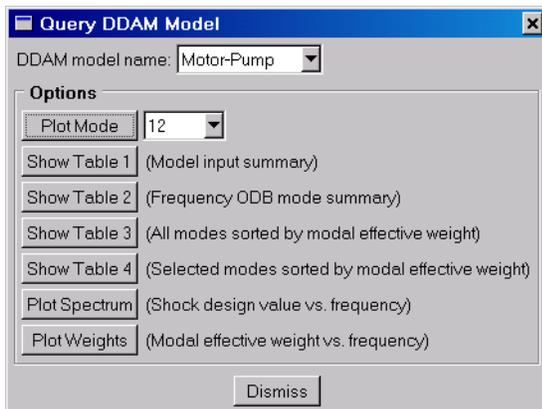

*Figure 12: **Query DDAM Model** dialog box.*

Create DDAM Job

A DDAM response spectrum analysis is initiated by creating and submitting a DDAM job. The **DDAM Job Manager**, shown in Figure 13, is used to manipulate DDAM jobs within the Abaqus/CAE architecture. DDAM for Abaqus places the DDAM response spectrum results into an output database that uses the DDAM job name as an identifier. Summary tables, modal weight, and SDV versus frequency plot files with the same identifier are generated automatically for easy inclusion into a DDAM report.
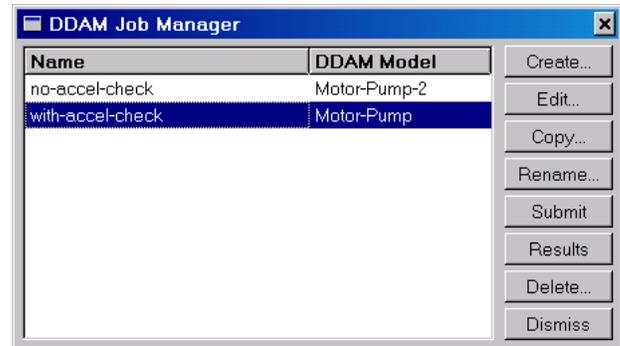

*Figure 13: **DDAM Job Manager**.*

Results for the Motor-Pump Example

The motor-pump example problem was run twice, with and without the nodal acceleration check mode selection criteria. Figure 14 shows the peak Mises stress results for all shell elements in the model for the SPOS location for the case where the nodal acceleration check was active.

Only shell stresses are shown in Figure 14 since DDAM for Abaqus places the DDAM peak results into separate variable name sets for the solid, shell, and beam elements. For shell elements, stress and strain output can be obtained at the top, middle, and bottom thickness locations. The peak Mises stress for the small cantilever panel under the motor is about 80 KSI. Figure 15 shows the same Mises stress plot for the case where the nodal acceleration check mode selection was not included. The peak response in Mises stress for this case is now less than 60 KSI, whereas the peak stresses in the other structural members show little or no change. This result illustrates the potential importance of the nodal acceleration check when evaluating the localized shock performance of light components within a larger finite element model.

Mode 12 of the frequency extraction was identified as the mode that was selected via the acceleration check that had a significant influence on the peak response of the cantilevered panel. The peak response of mode 12, as requested and stored on the DDAM response output database file, is shown in Figure 16, with the shape scaled by a factor of 100.

Conclusion

The DDAM for Abaqus custom GUI application provides a complete, convenient environment in which to perform DDAM analyses with Abaqus.
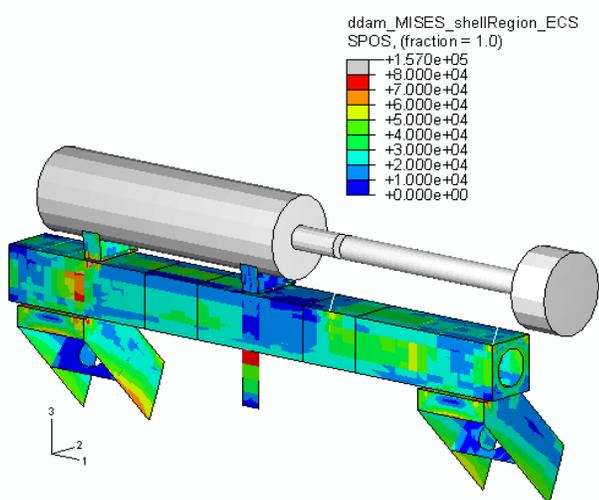
Figure 14: Peak Mises stress in shell elements with nodal acceleration check active.
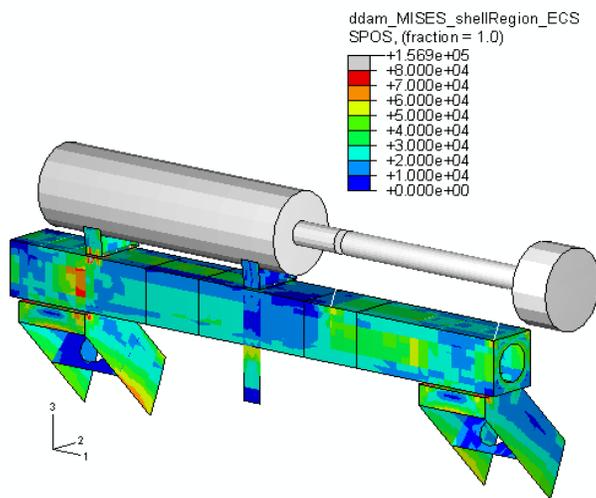
Figure 15: Peak Mises stress in shell elements without nodal acceleration check.
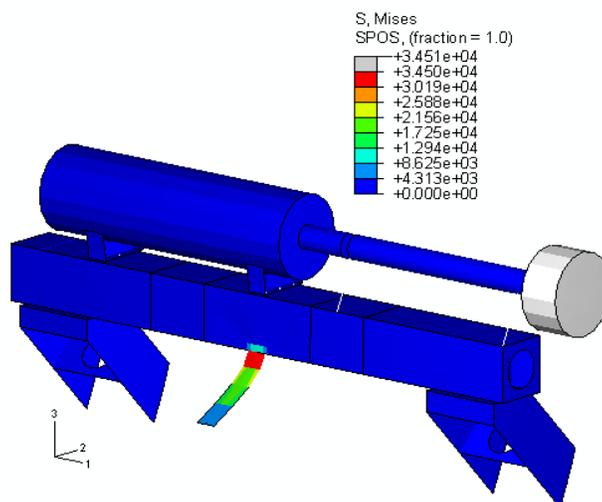
Figure 16: Peak Mises stress response in mode 12.

References

1. NAVSEA 0908-LP-000-3010, Rev. 1, "Shock Design Criteria for Surface Ships," Naval Sea Systems Command, September 1995.

2. O'Hara, G. J., R. O. Belsheim, "Interim Design Values for Shock Design of Shipboard Equipment," Naval Research Laboratory Report 1396, February 1963 (UNCLASSIFIED, available from the U.S. Department of Commerce, National Technical Information Service, Springfield, VA).

Abaqus References

To download DDAM for Abaqus, see Abaqus Answer 2373. For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.13 documentation references:

- Abaqus Analysis User's Guide
    - "Natural frequency extraction," Section 6.3.5
    - "Response spectrum analysis," Section 6.3.10
- DDAM for Abaqus: A User's Guide