Support Structure Superelement: User Guide for Bladed versions 4.8+

Author: Will Collier

Issue: K

Date of issue: 14 Dec 2020

Updates in this issue

- Title edited to no longer refer only to v4.8 & v4.9. Minor test edits also made for continuity

Doc no: 110052-UKBR-T-37-K

1 SUMMARY

This document summarises the theory and use of the superelement feature, available in Bladed 4.8 and later versions. This guide is written for Bladed 4.8.0.95 and later and is also applicable for Bladed 4.9 and later versions.

The superelement feature allows a reduced jacket model from detailed offshore structural modelling software (e.g. Sesam) to be imported into Bladed. The dynamic response of the wind turbine and jacket can be included in Bladed without an explicit model of the jacket defined in Bladed.

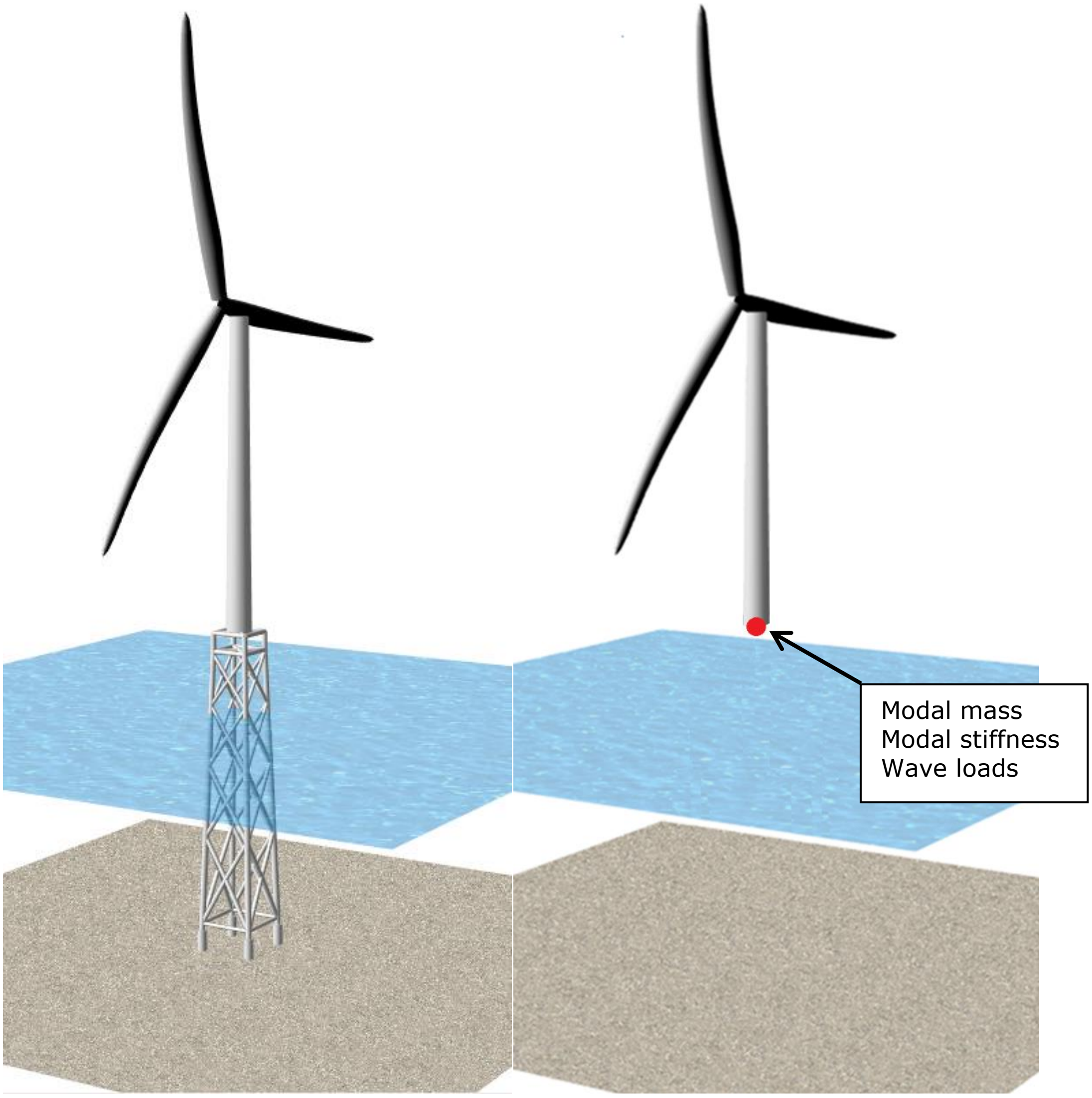


Figure 1: Superelement representation replaces jacket detail with modal information

2 SUPERELEMENT WORKFLOW

A superelement is a reduced (i.e. modal) model of a complex sub-structure like a jacket support structure. Superelement models of a jacket can be imported into Bladed for coupled analysis with the rest of the structure.

The basic workflow for using a superelement in Bladed is illustrated in Figure 2. Each image is explained by a bullet point below

- A complex jacket structure, comprising of any element type (e.g. shell and beam elements) is defined in an offshore modelling code, such as Sesam.
- The jacket structure is "reduced" to a modal model called a superelement. The superelement consists of a modal mass (M) and modal stiffness (K) matrix. A vector of wave loads (f) on each mode for each particular wave state is also stored with the superelement.
- The superelement model is imported into Bladed, and is included instead of the jacket model in Bladed. Coupled analysis of the jacket and turbine is performed. An accurate dynamic response of the coupled turbine and jacket structural system is captured thanks to the modal information contained in the superelement.
- The load time history from Bladed at the interface node between the tower and jacket is passed back to the offshore code. This load time series and the wave loads are applied to a model of the jacket in the offshore code.
- Fatigue and ultimate load checks are carried out in the offshore code.

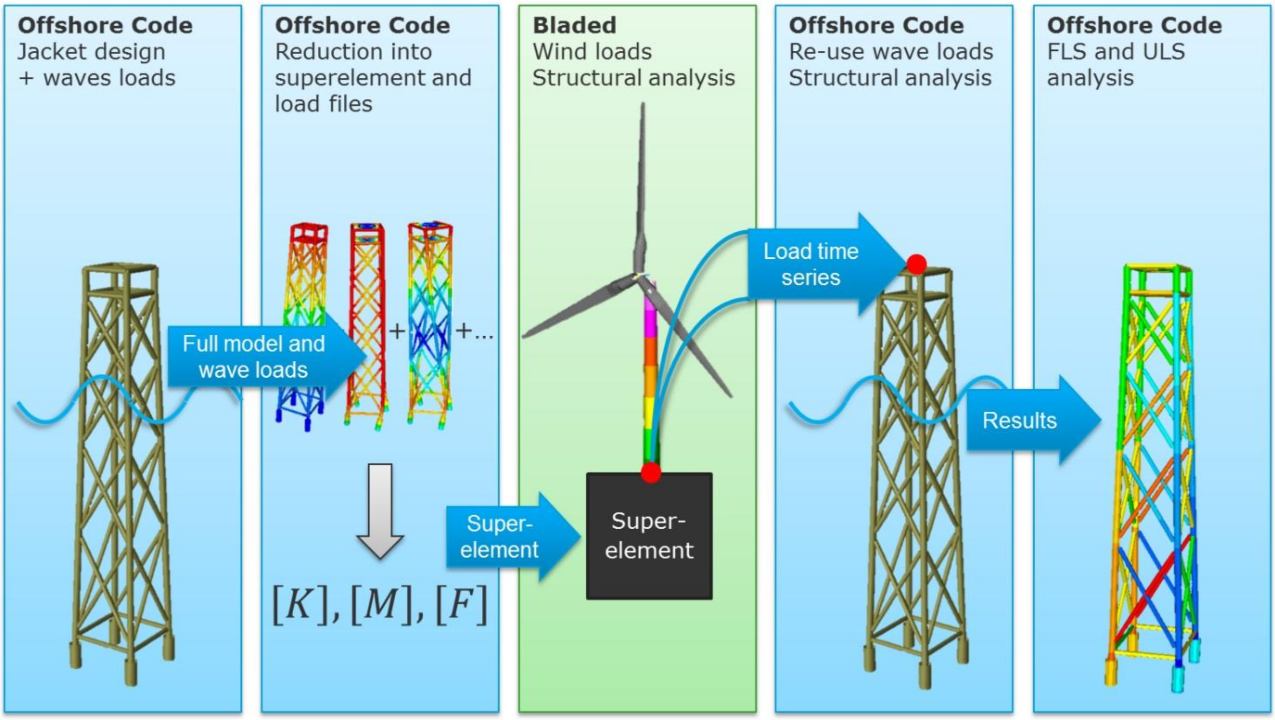


Figure 2: Superelement workflow with Bladed

The superelement analysis workflow can be compared to the Bladed workflow for fully integrated modelling, shown in Figure 3. In this case, the jacket model is defined in Bladed (or imported from an offshore code) and member loads are calculated directly in Bladed. Only the initial design of the jacket and post-processing of member loads is carried out in the offshore code.

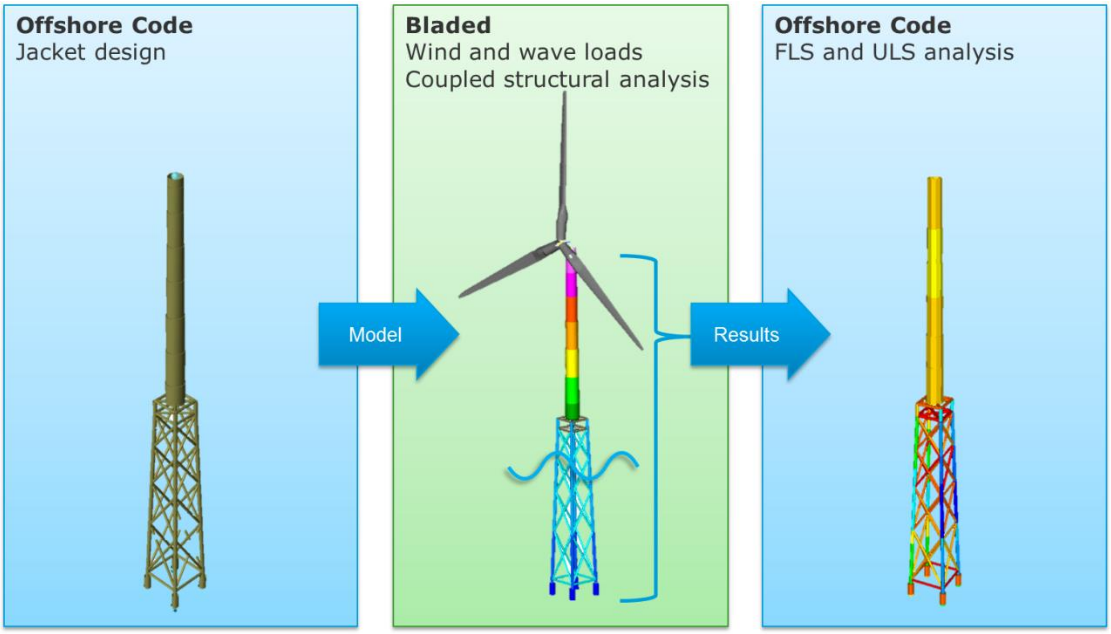


Figure 3: Integrated workflow in Bladed

The superelement workflow has two key **advantages** compared to a full integrated analysis in Bladed

- The foundation designer can share their design for aero-elastic analysis without needing to share the jacket design details with the wind turbine designer/manufacturer.
- Complex support structure elements (e.g. shell elements) can be included in the superelement. The dynamic response of the jacket including such elements can be included in the Bladed simulations.

However, there are also some **disadvantages** of adopting a superelement approach

- The hydro-elastic coupling between support structure and waves is not taken into account, as wave loads must be specified as a simulation input.
- The superelement is linear and cannot include features such as non-linear soil springs or geometric stiffness corrections.
- Time domain simulation is required in both Bladed and the offshore code in order to determine the interior jacket loads, which involves some duplication of simulation.
- Overall system optimisation is easier in a fully integrated model.

3 SUPERELEMENT MODEL CREATION

The finite element model of the jacket must be converted into reduced form and exported from the offshore modelling code e.g. Sesam. The Bladed superelement feature has been designed with Craig-Bampton [1] reduction for the superelement in mind. Craig-Bampton is the preferred method as an accurate dynamic response of the jacket can be retained in the superelement. Other reduction methods could in principle be used but are not considered in this document.

Craig-Bampton modes for the superelement consist of *constraint* and *normal* mode shapes. An important concept is division of the jacket nodes into the **b**oundary node and **i**nterior nodes. The boundary node is the node where the superelement connects to the tower base. The interior nodes are all of the other nodes in the jacket.

Constraint modes describe the displacement of the interior jacket nodes when six unit displacements (3 translational and 3 rotational) are applied at the superelement interface node (the boundary node). These mode shapes provide the static response of the superelement and allow the superelement interface node to move.

For the normal modes, the jacket structure is constrained at the base and the interface node and the eigenmodes are calculated. As the jacket is constrained at both ends, these are also referred to as **i**nterior modes (they include motion of the interior nodes only). The normal modes enhance the dynamic response of the superelement. The union of these two sets of modes can provide an accurate dynamic model of the jacket and motion of the interface node.

The outputs of the superelement creation process are a mass matrix and stiffness matrix for the reduced model. A wave load time history for each mode is also output. A reduction basis $[R]$, based on the constraint and normal mode shapes, is used to reduce the jacket finite element matrices as follows

$$[\bar{M}] = [R]^T [M_{FE}][R] \tag{1}$$

$$[\bar{K}] = [R]^T [K_{FE}][R] \tag{2}$$

$$\bar{f} = [R]^T \begin{bmatrix} f_b \\ f_i \end{bmatrix} \tag{3}$$

where

$[R]$	reduction basis that transforms the jacket FE model into a superelement model
$[M_{FE}]$	finite element mass matrix
$[K_{FE}]$	finite element stiffness matrix
$[\bar{M}]$	superelement mass matrix
$[\bar{K}]$	superelement stiffness matrix
f_b	external applied wave load at the **b**oundary node
f_i	external applied wave load at the **i**nterior nodes
$\bar{f}$	superelement external applied wave load

It's important to note that the first six degrees of freedom in the superelement model correspond to displacement of the interface node (boundary node). The remaining degrees of freedom correspond to the interior normal modes. Sufficient normal modes should be included to obtain an accurate dynamic response. The reduced equations of motion for the superelement are expressed as follows

$$[\bar{M}] \begin{bmatrix} \ddot{x}_b \\ \ddot{\eta}_i \end{bmatrix} + [\bar{K}] \begin{bmatrix} x_b \\ \eta_i \end{bmatrix} = \bar{f} \tag{4}$$

where

x_b	the six **b**oundary DoF displacements (equivalent to constraint mode amplitudes).
η_i	amplitude of each normal mode (*i* stands for **i**nterior)

Further details of a typical Craig-Bampton reduction basis are given in Appendix A.

3.1 Superelement mode shape matrix

The reduced matrices $\bar{M}$ and $\bar{K}$ include all the necessary properties to describe the dynamics of the superelement in Bladed. However, it is also necessary for Bladed to know how the interface node in the Bladed structural definition couples to the superelement modal displacements i.e. how this interface node moves when each superelement degree of freedom is activated.

By default, a mode shape matrix for the superelement is assumed by Bladed as shown below. The 6 boundary degrees of freedom are assumed to correspond to translational and rotational motion in the Bladed global coordinate system. The normal mode amplitudes do not cause motion of the interface node. However, note that the normal modes do contribute to the dynamic response of the superelement due to off diagonal terms in the superelement mass and stiffness matrices.

$$\begin{bmatrix} x_{interface\_x\_disp} \\ x_{interface\_y\_disp} \\ x_{interface\_z\_disp} \\ x_{interface\_x\_rot} \\ x_{interface\_y\_rot} \\ x_{interface\_z\_rot} \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & . & . \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & & \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & & \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & & \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & & \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & & \end{bmatrix} \begin{bmatrix} x_{b1} \\ x_{b2} \\ x_{b3} \\ x_{b4} \\ x_{b5} \\ x_{b6} \\ \eta_{i1} \\ \eta_{i2} \\ \eta_{i3} \\ \eta_{i4} \\ . \\ . \end{bmatrix} \qquad (5)$$

In the case that the superelement has been generated in a coordinate system different from that used in Bladed, an additional transformation is required, as described in section 6.1.1.

4 SUPERELEMENT FILE FORMAT

For superelement runs in Bladed, two external input files are required: the superelement description file and the wave loads file. Bladed currently supports the superelement format exported by DNV GL's Sesam and Ramboll's ROSA software. This format matches the format used in the Siemens aero-elastic code BHAWC.

4.1 Superelement description file

The superelement description file consists of the reduced mass, stiffness and damping matrices and a gravity load vector. An example can be viewed in the text file below, for a superelement with 7 degrees of freedom.

superelement_description_file.txt

Note that the mass, stiffness and damping matrices are square and are sized by the number of modes. The gravity load vector specifies a constant gravity load per mode if desired.

It is essential to include some structural damping on the superelement to achieve physically realistic results in time domain simulations. One way to do this is Rayleigh damping [2], which is calculated by multiplying the mass and stiffness matrices by constant values.

$$[C] = a_0[K] + a_1[M] \tag{6}$$

The damping can be calculated in the offshore code (e.g. Sesam) or can be added manually to the superelement description file.

4.2 Superelement wave loads file

The wave load file primarily consists of the external load applied to each superelement degree of freedom, corresponding to a particular sea state. An example can be viewed in the text file below for a superelement with 7 degrees of freedom (6 constraint modes and one normal mode). Note that there is an additional final column for the sea surface elevation but this is not currently used by Bladed.

wave_loads_file.txt

The wave loads can be read by Bladed as N or kN by selecting the desired option in the superelement definition screen (see section 6.1).

5 SUPERELEMENT DAMPING DEFINITION

The integrated and superelement approaches use a different structural mode basis, as illustrated in Figure 4. In an integrated modes, modes cover the entire support structure, whereas in the superelement approach, separate mode shapes are defined for the superelement and tower modes.

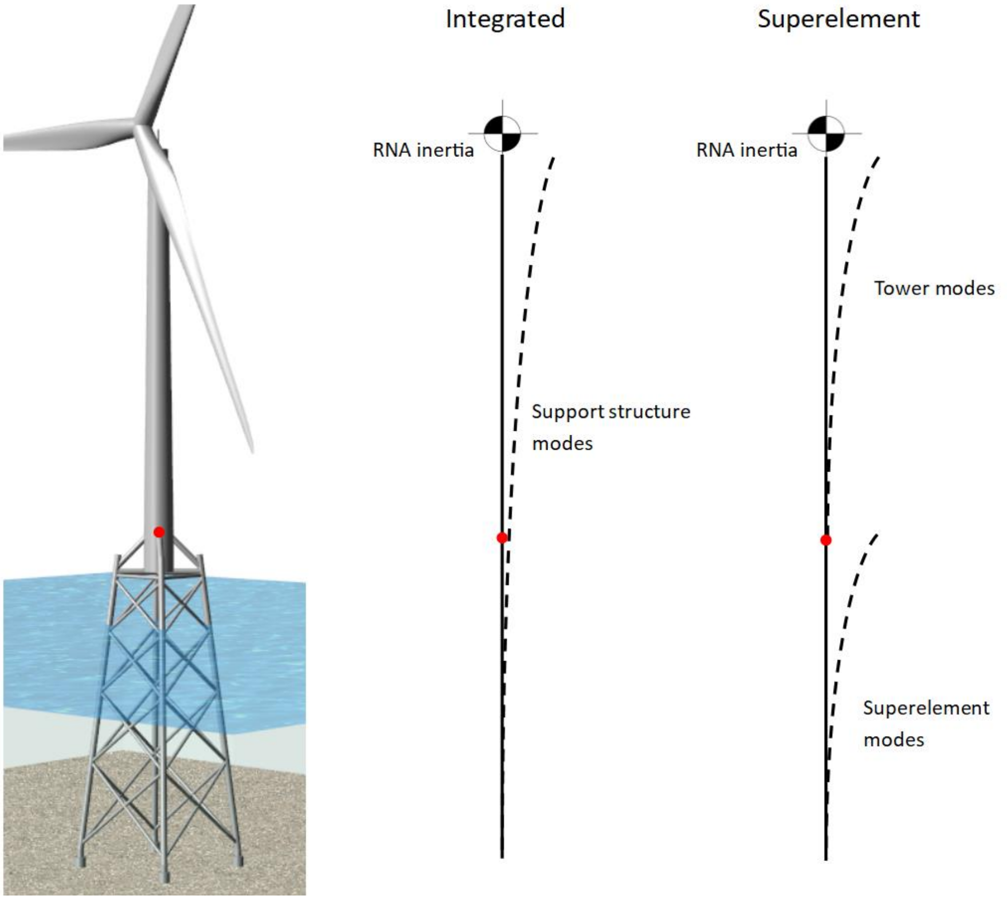


Figure 4: Modal basis in integrated and superelement approaches

In Bladed, damping is defined on the modal degrees of freedom. This leads to the question *how can equivalent damping be defined in the integrated and superelement approaches?*

In this section, the methods and challenges for damping definition are discussed for both integrated and superelement approaches. A method is then described that overcomes these difficulties and allows equivalent damping to be defined on integrated and superelement approaches.

5.1 Nomenclature

The following symbols and abbreviations are used in this section.

SE	Superelement (representing jacket)
T	Tower
SS	Support structure (jacket and tower together)
RNA	Rotor nacelle assembly
$M_{SS,comp}$	Modal mass matrix, SS component modes
$K_{SS,comp}$	Modal stiffness matrix, SS component modes
$C_{SS,comp}$	Modal damping matrix, SS component modes
$M_{T,comp}$	Modal mass matrix, T component modes
$K_{T,comp}$	Modal stiffness matrix, T component modes
$C_{T,comp}$	Modal damping matrix, T component modes
$M_{SE,comp}$	Modal mass matrix, SE component modes
$K_{SE,comp}$	Modal stiffness matrix, SE component modes
$C_{SE,comp}$	Modal damping matrix, SE component modes
$M_{SS,nat}$	Modal mass matrix, SS natural modes
$K_{SS,nat}$	Modal stiffness matrix, SS natural modes
$C_{SS,nat}$	Modal damping matrix, SS natural modes
$\Psi_{towertop}$	Tower top node mode shape matrix (6x6)
M_{RNA}	RNA inertia matrix (6x6)
$M_{i,rotor}$	Modal inertia associated with mode *i*
ζ_i	Modal damping ratio for mode *i*
ω_i	Modal angular frequency for mode *i*
ψ_i	Mode shape vector for mode *i*

5.2 Integrated model damping

For an integrated model, typically modal damping ratios are defined on the uncoupled Craig Bampton vibration support structure modes.

The modes are used to calculate modal stiffness and mass matrices for the support structure. The structure of the modal mass and stiffness matrices are shown in equation (7). The stiffness matrix is diagonal but the mass matrix is not.

$$[M_{SS,comp}] = \begin{bmatrix} M_{SS11} & M_{SS12} & M_{SS13} & . \\ M_{SS21} & M_{SS22} & M_{SS23} & \\ M_{SS31} & M_{SS32} & M_{SS33} & \\ . & & & . \end{bmatrix} \tag{7}$$

$$
[K_{SS,comp}] = \begin{bmatrix} K_{SS11} & 0 & 0 & . \\ 0 & K_{SS22} & 0 & \\ 0 & 0 & K_{SS33} & \\ . & & & . \end{bmatrix}
$$

where subscript SS is short for support structure.

In order to calculate the frequency of each mode, the RNA inertia associated with each attachment mode must also be taken into account. For each attachment mode, a quantity $M_{i,rotor}$ is defined as follows

$$
M_{i,rotor} = \Psi^{T}_{\text{towertop}}\, [M_{RNA}]\, \Psi_{\text{towertop}} \tag{8}
$$

where $[M_{RNA}]$ is the 6x6 RNA inertia matrix
Ψ_{towertop} is the mode shape matrix for the tower top node only

The modal angular frequency ω_i is then calculated as

$$
\omega_i = \sqrt{\frac{K_{SS,comp,ii}}{M_{SS,comp,ii} + M_{i,rotor}}} \tag{9}
$$

where $M_{i,rotor}$ is non-zero for each attachment mode and zero for the normal modes.

The support structure modal damping matrix is the calculated as shown in equation (10).

$$
[C_{SS,comp}] = 2 \begin{bmatrix} \frac{\zeta_1 K_{SS11}}{\omega_1} & & & \\ & \frac{\zeta_2 K_{SS22}}{\omega_2} & & \\ & & . & \\ & & & . \end{bmatrix} \tag{10}
$$

where

ζ_i are the modal damping ratios
ω_i are modal angular frequencies (rad/s)
K_{SSii} are the diagonal terms of the modal stiffness matrix

5.3 Superelement model damping (Bladed 4.8)

In Bladed 4.8, damping is defined separately on the tower and superelement, which each have their own mode shapes and damping definitions.

For the superelement, typically Rayleigh damping parameters are available based on the superelement mass and stiffness matrices.

$$[C_{SE,comp}] = a_0[M_{SE,comp}] + a_1[K_{SE.comp}] \tag{11}$$

The tower modal damping matrix will take a similar form to that for the integrated approach.

$$[C_{T,comp}] = 2\begin{bmatrix} \frac{\zeta_1 K_{T11}}{\omega_1} & & & \\ & \frac{\zeta_2 K_{T22}}{\omega_2} & & \\ & & \cdot & \\ & & & \cdot \end{bmatrix} \tag{12}$$

These matrices are simply combined to give a support structure damping matrix for the following format.

$$[C_{SS,comp}] = \begin{bmatrix} [C_{SE,comp}] & [\quad 0 \quad] \\ [\quad 0 \quad] & [C_{T,comp}] \end{bmatrix} \tag{13}$$

where subscript SS is short for support structure
subscript SE is short for superelement
subscript T is short for tower

There are two potential problems with this approach.

Firstly, the "cross terms" between the tower and superelement components are zero. This means that it is not possible with this approach to specify damping in a Bladed superelement model that is exactly equivalent to damping values that are specified on whole support structure modes.

Secondly, the supplied superelement damping matrix does not typically account for the effect of the inertia of the tower and rotor nacelle assembly (RNA).

5.4 Unified damping for superelement and integrated methods (Bladed 4.9+)

It is shown in section 5.1 and 5.3 that the described damping methods are not equivalent, for the following reasons

1. Damping is specified on a different set of modes for the superelement and integrated approaches

2. The superelement damping method does not include cross terms between the tower and superelement damping

3. The superelement damping approach does not take into account the inertia of the tower and RNA when calculating the superelement damping

To unify the damping for the superelement and integrated approaches, it is proposed to define the damping on a set of modes that is common to both approaches i.e. the coupled vibrational modes for the support structure. If the structural properties of the superelement are defined in a valid way, then the support structure coupled modes will be very similar to the integrated case.

5.4.1 Theory basis

The aim is to specify damping on the support structure coupled modes, and then transform this damping onto the actual degrees of freedom for the tower and superelement which are used in the simulation. This method is based on theory presented in reference [2].

Consider the partitions of the system mass and stiffness matrices relating to the support structure degrees of freedom. For the superelement approach, the uncoupled support structure mass and stiffness matrices have the form shown in equation (14) and (15).

$$\left[M_{SS,comp}\right] = \begin{bmatrix} \left[M_{SE,comp}\right] & \left[\quad 0 \quad\right] \\ \left[\quad 0 \quad\right] & \left[M_{T,comp}\right] \end{bmatrix} \tag{14}$$

$$\left[K_{SS,comp}\right] = \begin{bmatrix} \left[K_{SE,comp}\right] & \left[\quad 0 \quad\right] \\ \left[\quad 0 \quad\right] & \left[K_{T,comp}\right] \end{bmatrix} \tag{15}$$

For the integrated approach, the same matrices have a more simple form, based on the form shown in equation (7), as shown in equations (16) and (17).

$$\left[M_{SS,comp}\right] = \begin{bmatrix} \left[\quad M_{SS} \quad\right] \end{bmatrix} \tag{16}$$

$$\left[K_{SS,comp}\right] = \begin{bmatrix} \left[\quad K_{SS} \quad\right] \end{bmatrix} \tag{17}$$

Note that $M_{SS,comp}$ and $M_{T,comp}$ include the influence of the rigid RNA inertia on each tower attachment mode. For the purpose of the damping calculation, M_{SE} is edited to include the influence of the tower and the rigid RNA on each constraint mode, using a method equivalent to the formulation in equation (8).

The coupled mode shapes for the support structure are found by solving the structural eigen problem using the uncoupled mode shape matrices. The number of coupled mode shapes for the support structure is equal to the sum of the number of tower and superelement modes.

$$\left[K_{SS\ comp}\right] \psi_i = \omega_i^2 \left[M_{SS\ comp}\right] \psi_i \tag{18}$$

where

ψ_i are the mode shape vectors

ω_i are modal angular frequencies (rad/s)

The assembled square mode shape matrix $[\Psi]$, where each column holds an individual mode shape ψ_i, describes coupled mode shapes in terms of the contributions from the uncoupled mode shapes. The mode shape matrices are used to transform uncoupled properties to coupled properties:

$$\left[K_{SS,nat}\right] = \left[\Psi^T\right]\left[K_{SS,comp}\right]\left[\Psi\right] \tag{19}$$

$$[M_{SS,nat}] = [\Psi^T][M_{SS,comp}]\,[\Psi]$$

Damping is specified on the coupled modes using proportional or modal damping

Proportional damping: $$[C_{nat}] = a_0[M_{nat}] + a_1[K_{nat}] \quad (20)$$

Modal damping: $$[C_{nat}] = 2\begin{bmatrix} \frac{\zeta_1 K_{nat_{11}}}{\omega_1} & & & \\ & \frac{\zeta_2 K_{nat_{22}}}{\omega_2} & & \\ & & \cdot & \\ & & & \cdot \end{bmatrix} \quad (21)$$

where

a_0 and a_0 are proportionality constants
ζ_i are modal damping ratios
ω_i are modal angular frequencies (rad/s)

This damping on the coupled modes is then transformed back onto the uncoupled modes for use in the simulation

$$[C_{SS,comp}] = [\Psi^T]^{-1}[C_{SS,nat}]\,[\Psi]^{-1} \quad (22)$$

$C_{SS,nat}$ is a full matrix that has coupling terms between the tower and superelement components (for a superelement model), or coupling between the support structure modes (for and integrated model). $C_{SS,nat}$ also includes the influence of the RNA inertia.

5.4.2 Results comparison

The effect of specifying damping on the support structure coupled modes is demonstrated in Figure 5.

The coupled mode damping ratios for an integrated and superelement model are compared, using a jacket support structure from study in reference [4] with the RNA included. The support structure coupled mode frequencies are derived by running a Campbell diagram calculation with rigid RNA. The target damping ratio is 0.5% on the first two modes, and 1.0% on the second two modes.

The orange bars show the damping ratios when damping is defined separately on the tower and superelement modes, as in section 5.3. The grey and blue bars show the damping ratios for integrated and superelement approaches with damping defined on the support structure coupled modes. It is seen that defining the damping on support structure coupled modes results in equivalent damping for the two approaches up to 5Hz.

Definition of damping on the uncoupled superelement and tower modes results in incorrect damping on the first two modes. In section 5.5 it is discussed how the uncoupled mode damping can be tuned to give the desired damping ratio for the first few coupled modes.

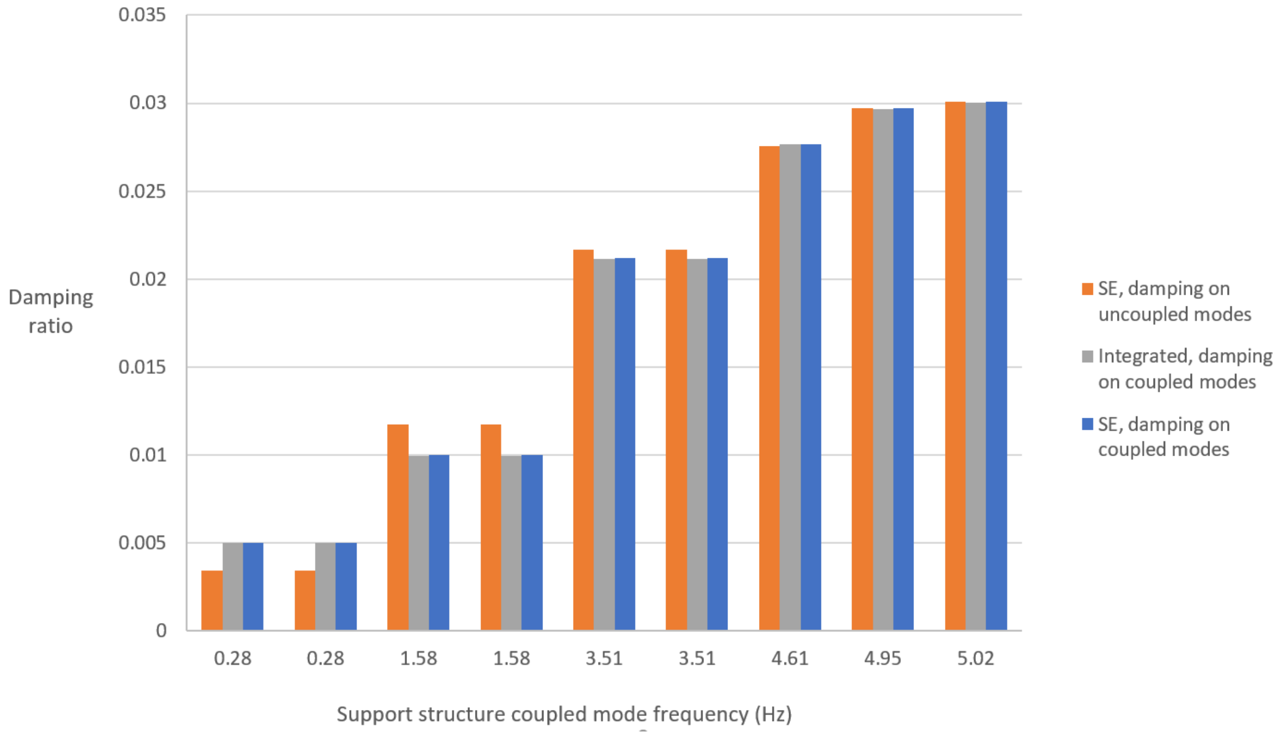


Figure 5: Support structure coupled mode damping ratios for integrated and superelement models, including effect of specifying damping on the support structure coupled modes

5.5 Specifying superelement damping in 4.8

In Bladed 4.9 and later, it is recommended to specify damping on the support structure coupled modes, as discussed in section 5.4. However, this functionality is not available in Bladed 4.8, so for this version a different method must be used to achieve the target damping for the support structure modes when using a superelement method.

In Bladed 4.8, the uncoupled tower modal damping and superelement damping specified by the user are used directly in the calculation, in the form of equation (13). This simple approach means that it is not straightforward to achieve equivalent damping for an integrated structure and superelement structure, as shown in section 5.4.2.

The target damping ratios for the first few support structure coupled modes can be achieved by tuning the damping on the superelement and/or tower. A possible strategy for such a tuning exercise is:

- Define target damping values for the desired support structure coupled modes
- For each support structure coupled mode where the damping needs to be adjusted, examine the contributions of the uncoupled modes using a Campbell diagram calculation for a parked turbine and with a rigid rotor
- Raise or lower the damping on the uncoupled modes that are important in the coupled mode in order to adjust towards the target damping ratio

This method typically requires some iteration to achieve the desired damping on the first few support structure coupled modes. It can be necessary to adjust both the superelement and tower damping values.

Example results of this tuning exercise are illustrated in Figure 6, using the same model discussed in section 5.4.2. It observed in the blue bars that it was possible to achieve the target damping on the first two support structure coupled modes. However, higher modes, such as at 3.51Hz do not achieve the target damping ratio.

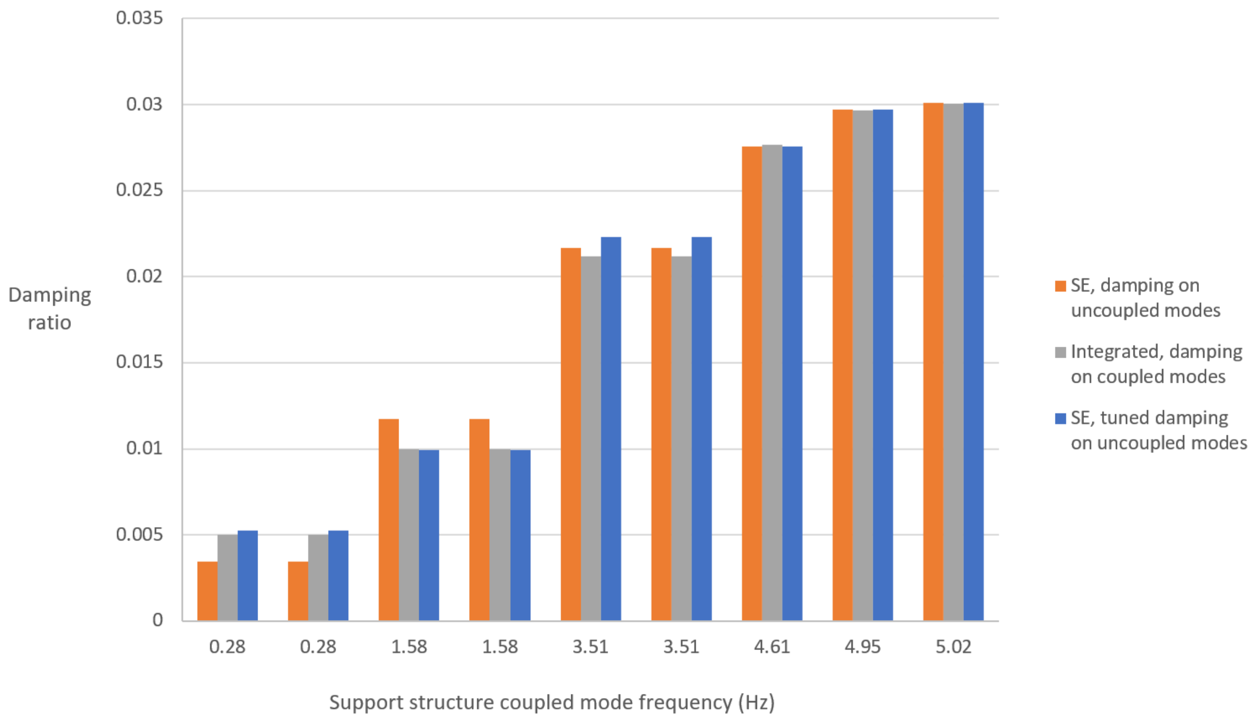


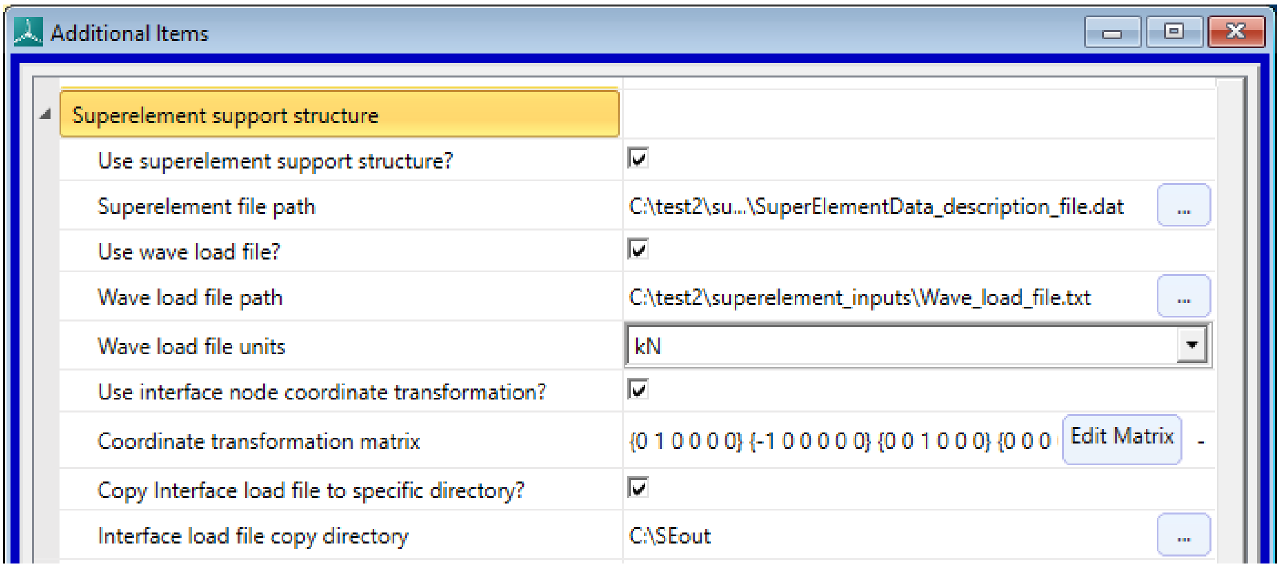
Figure 6: Support structure coupled mode damping ratios for integrated and superelement models, including effect of tuning the damping for the superelement model

6 SUPERELEMENT DEFINITION IN BLADED

From Bladed 4.8.0.88, superelement definition is supported in the Bladed GUI. Note that the Bladed "Offshore support structure" licence is required to use a superelement.

6.1 Superelement screen

The superelement properties can be defined in the *Additional Items* screen, as shown below. The path to the superelement file and wave load file can be set. The wave load file is optional.



6.1.1 Superelement coordinate system

If the superelement has been generated in a coordinate system that differs from the Bladed global coordinate system, then it is necessary to use the "Use interface node coordinate transformation" option.

If this option is disabled, Bladed will assume that the superelement was created according to the Bladed global coordinate system i.e. that the first 6 modes are constraint modes corresponding to unit displacements of Xdef, Ydef, Zdef, Xrot, Yrot, Zrot and the remaining modes are normal modes with no associated interface motion.

However, if the superelement is generated using a different coordinate system then it will be necessary to specify the transformation from the superelement coordinate system to the Bladed global coordinate system. An example coordinate system difference is shown in Figure 7. This transformation could be achieved by specifying the following matrix.

0	1	0	0	0	0
-1	0	0	0	0	0
0	0	1	0	0	0
0	0	0	0	1	0
0	0	0	-1	0	0
0	0	0	0	0	1

Note that, if defined, this matrix replaces the first 6x6 entries in the mode shape matrix given in section 3.1, equation (5) to ensure that the superelement boundary degrees of freedom correspond to the appropriate motion of the interface node.

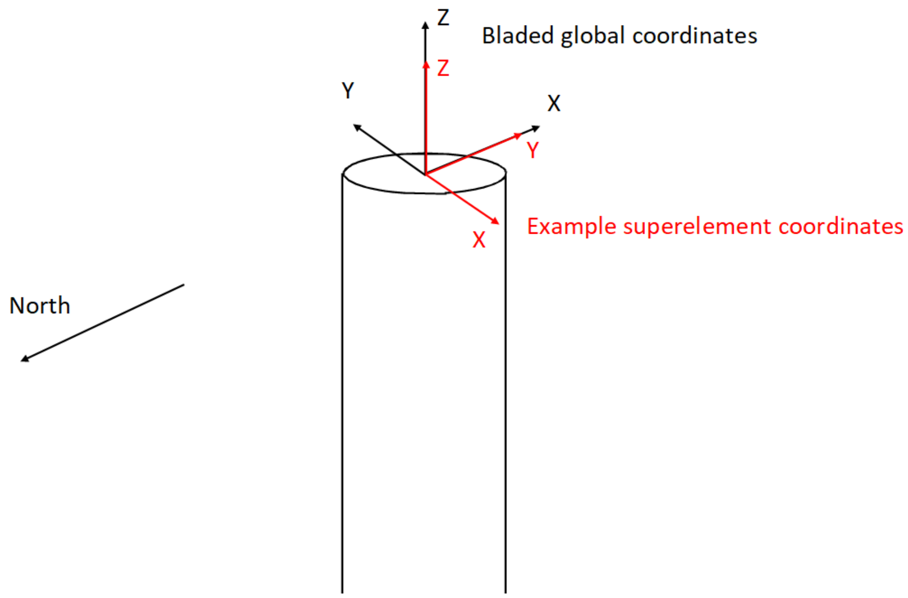


Figure 7: Bladed global coordinate system and possible coordinate system used in code that generated the superelement

6.2 Support structure screen settings

In the support structure screen, the follow settings should be observed:

- A **multi-member tower** type must be selected.
- An **onshore** turbine is usually selected as the wave loads are applied to the superelement.
- Define the tower nodes and members corresponding to the tower.
- The superelement should connect to the tower at the lowest tower node. At this node, a **Rigid** foundation node should be added. There must be exactly one foundation node at the tower base. This is illustrated in Figure 8.

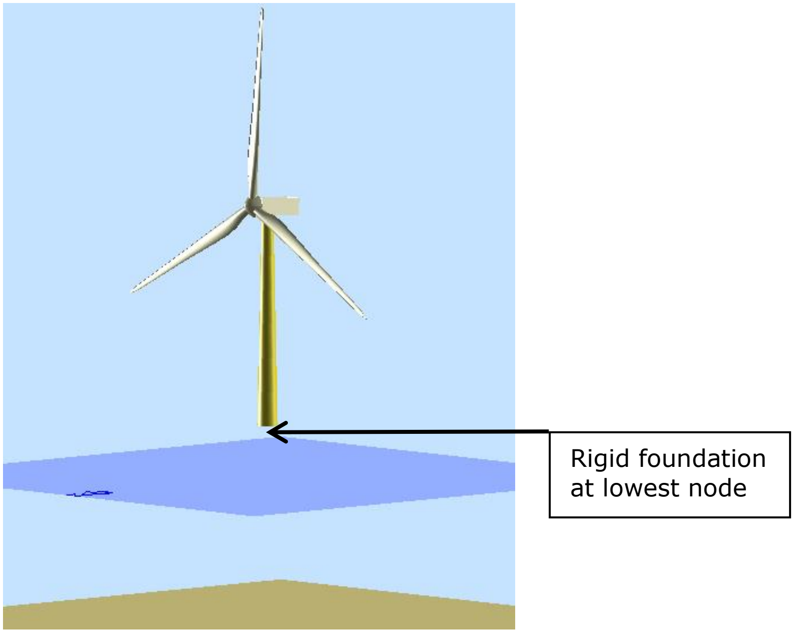


Figure 8: Tower definition for superelement simulation

6.3 Damping definition

6.3.1 Bladed 4.8

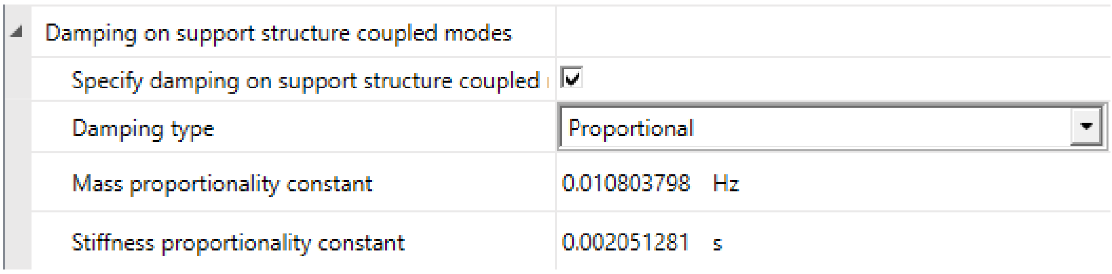
In Bladed 4.8, damping is specified separately on the superelement (in the superelement description file) and on the tower component (in the Flexibility Modeller screen).

6.3.2 Bladed 4.9+

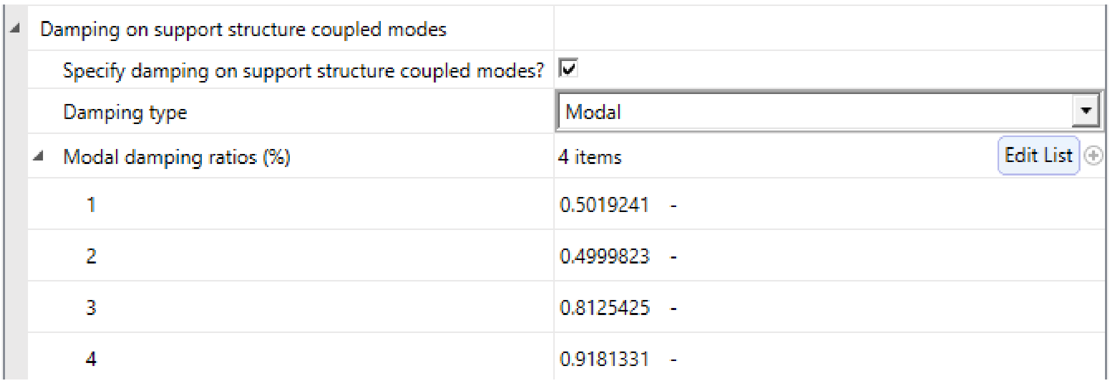
In Bladed 4.9, and later versions, it is recommended to specify superelement damping on the coupled modes of the support structure (although the method from section 6.3.1 is also supported).

Damping on support structure coupled modes can be specified in the *Additional Items* screen. If damping is specified in this manner, then the tower uncoupled model damping and the damping in the superelement description file are not used.

For proportional damping, the proportionality constants must be specified:



For modal damping, the damping ratios are specified for as many modes as desired:



Modal damping ratios of higher frequency modes will be calculated automatically by Bladed, based on the damping ratio of the highest frequency mode for which damping is defined, using the following relationship

$$\zeta_i = \zeta_c \left(\frac{\omega_i}{\omega_c}\right)$$

where

ζ_c and ω_c are the modal damping ratio and angular frequency for the last mode with damping defined

ζ_i and ω_i are the modal damping ratio and angular frequency for each higher frequency coupled mode

Note that the support structure coupled mode frequencies (assuming a rigid RNA) are output to the $VE file when any simulation is run with the “specify damping on support structure coupled modes?” feature is active. Search for the string “`DAMPING ON SUPPORT STRUCTURE COUPLED MODES`” in the $VE file to find this information, and to view the system damping matrix before and after the damping transformation.

6.4 Integrator settings

Typically the superelement will contain many degrees of freedom with some at high natural frequency. It may therefore be beneficial to use a Newmark Beta or Generalised Alpha integrator to achieve good simulation performance. This can be selected in the *Integrator settings* screen.

6.5 Simulation outputs

6.5.1 Interface load outputs

There is a special output group for superelement simulations that gives the superelement interface node kinematics and loads. This is the group *Superelement interface outputs*.

6.5.2 Interface load file

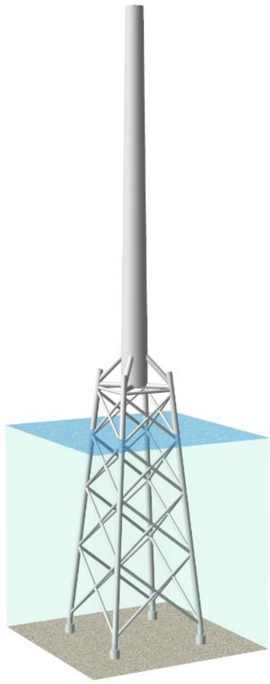
A file containing the interface node internal loads is also output with the name in the format *wave_loads_file_name.interface*. This file can be read directly by support structure codes like Sesam for post-processing.

From Bladed 4.8.0.95, the interface load files can be copied to a common location so that they are gathered together for post-processing. This option can be enabled in the Superelement GUI in the Additional Items screen. Note that because the interface load files are named after the wave load files, all wave load files should have unique names when using this option.

7 VERIFICATION

The superelement functionality has been validated through simulations using Sesam and Bladed. Full details can be found in the document "Verification report of Sesam's Bladed interface" on the Bladed portal. A brief summary of some of the work is given in this section.

7.1 Introduction



Simulations were performed for an offshore jacket as used in the DNV GL FORCE project [3].

The jacket and tower were defined in offshore code Sesam, and exported for use in Bladed. A superelement model of the jacket was also exported from Sesam.

This allowed definition of three numerical models for comparison

- "Sesam integrated" model of jacket and tower
- "Bladed integrated" model of jacket and tower
- "Bladed superelement" model with tower in Bladed and jacket in superelement from Sesam

7.2 Modal frequency comparison

The natural vibration frequencies were compared for between the "Sesam integrated" and "Bladed superelement" models, as shown in Table 7-1. An excellent agreement is found with a maximum error of 0.16%.

Mode	Sesam integrated (Hz)	Bladed superelement (Hz)	% error
1	0.281	0.281	0.00
2	0.281	0.281	0.06
3	1.578	1.579	-0.03
4	1.578	1.579	-0.03
5	3.513	3.508	-0.16
6	3.513	3.508	-0.16
7	4.606	4.609	0.01
8	4.955	4.955	-0.01
9	5.015	5.016	0.00
10	5.398	5.400	-0.01
11	5.398	5.400	-0.01
12	6.180	6.180	-0.01
13	6.425	6.425	-0.01
14	6.592	6.592	0.00
15	6.864	6.877	-0.05

Table 7-1: Comparison of jacket natural frequencies for "Sesam integrated" and "Bladed superelement" models

7.3 Time domain comparison

Equivalent irregular sea states (Hs = 4.8m, Tp = 8.6s) were defined in Bladed and Sesam and used to apply loads to the jacket models. The wave direction was aligned with the global x-direction.

Figure 9 shows the comparison of interface loads between the three methods. Figure 10 shows the interface node displacements. In both cases, a near perfect agreement is shown between the "Bladed superelement" and "Sesam integrated" results. The "Bladed integrated" run shows slightly different results due to differences in the calculation of the applied wave loads.

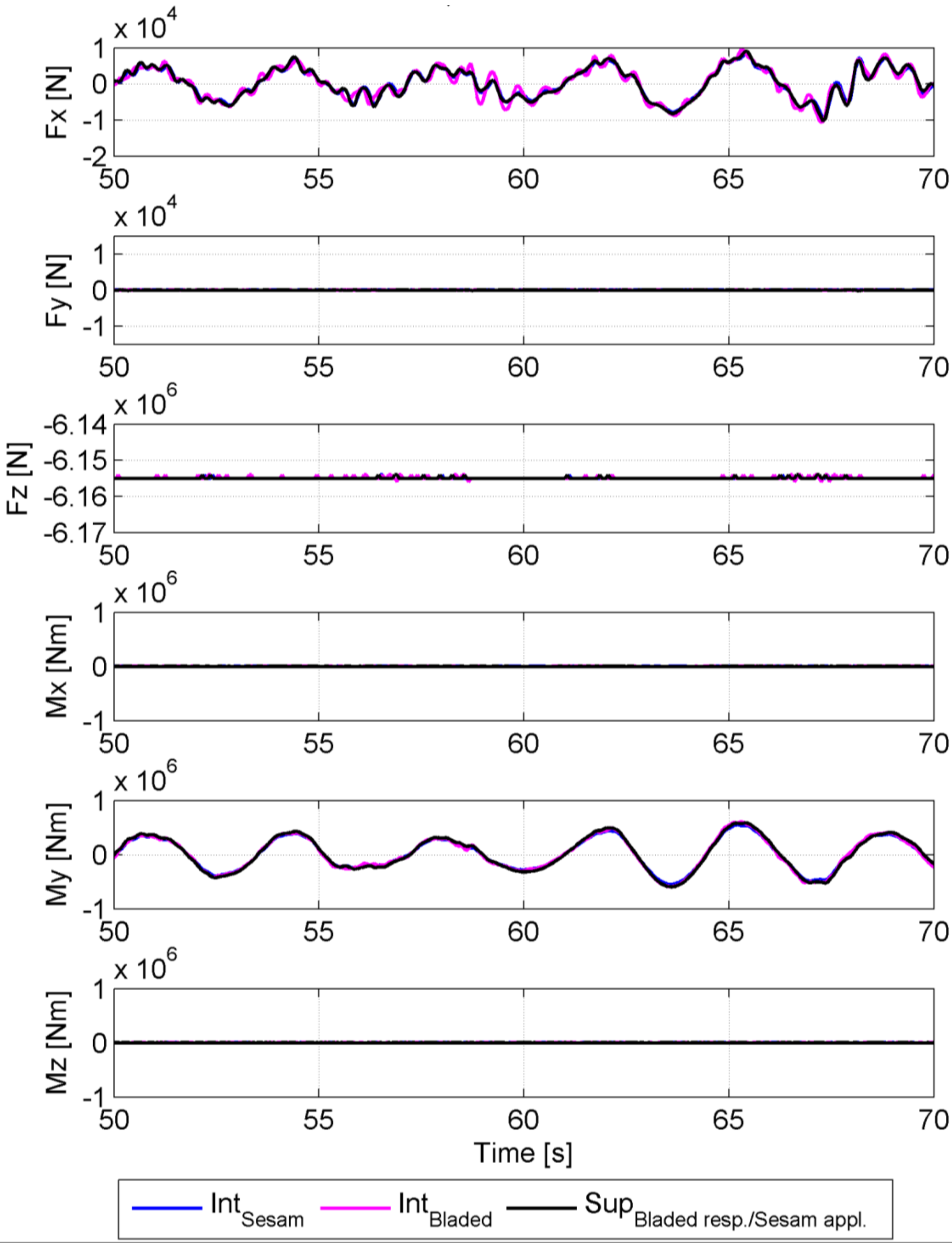


Figure 9: Interface load comparison

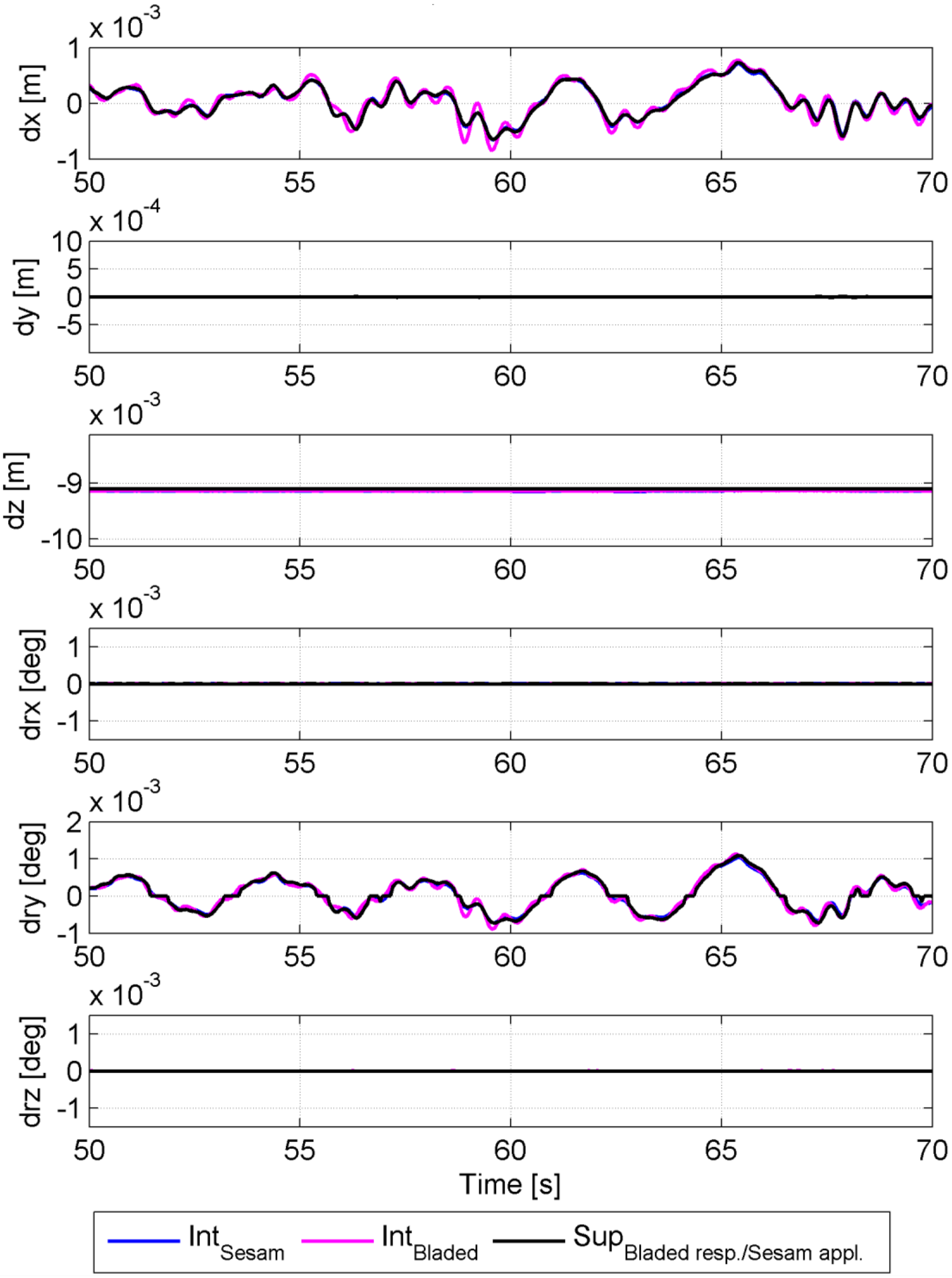


Figure 10: Interface node displacements

8 REFERENCES

1. Craig R, "*Coupling of substructures for dynamics analyses: an overview*" AIAA-2000-1573
2. Clough, Penzien, "*Dynamic of structures*" Second Edition. pp 234-235
3. https://www.dnvgl.com/news/integrated-approach-to-wind-turbine-structure-will-reduce-cost-of-offshore-wind-by-at-least-10--8043
4. DNV GL report "*Verification report of Sesam's Bladed interface*" No. 2016-0866

APPENDIX A: CRAIG-BAMPTON REDUCTION BASIS

The aim of the reduction is to reduce the number of degrees of freedom in the jacket structure and export the resulting matrices in a compact form ready for import to Bladed for dynamic analysis.

To perform the Craig-Bampton reduction, the finite element model equations are expressed with the boundary and interior nodes in matrix partitions

$$\begin{bmatrix} M_{bb} & M_{ib} \\ M_{bi} & M_{ii} \end{bmatrix} \begin{bmatrix} \ddot{x}_b \\ \ddot{x}_i \end{bmatrix} + \begin{bmatrix} K_{bb} & K_{ib} \\ K_{bi} & K_{ii} \end{bmatrix} \begin{bmatrix} x_b \\ x_i \end{bmatrix} = \begin{bmatrix} f_b \\ f_i \end{bmatrix} \tag{23}$$

where subscript *i* refers to interior nodes, and *b* for boundary nodes. All nodes are interior nodes except for the superelement interface node, which is at the boundary of the superelement. Note that damping is excluded at this stage and can be added directly to the reduced model.

The motion of the boundary nodes will be described by constraint modes, which correspond to unit displacements and rotations of the boundary node. The boundary node is retained in the reduction method.

To perform the reduction, it is desirable to express the motion of the interior nodes in terms of the static and dynamic response.

$$x_i = x_{i,stat} + x_{i,dyn} \tag{24}$$

The static response of the interior nodes is described by constraint modes. Constraint modes describe the displacement of the interior nodes based on the displacement of the boundary nodes.

$$x_{i,stat} = -K_{ii}^{-1} K_{ib} x_b = \Psi_{\mathrm{ib}} x_b \tag{25}$$

Ψ_{ib} is the matrix of constraint modes.

The dynamic response of the interior nodes is captured by normal modes calculated by solving the eigen problem for the interior nodes

$$\left[K_{ii} - \omega_j^2 M_{ii}\right]\{\phi_i\}_j = 0 \tag{26}$$

A chosen number of normal modes *m* are retained, and collected in a matrix of normal modes

$$\Phi_{\mathrm{i}} = [\phi_{i1}, \phi_{i2}, \ldots\ldots, \phi_{im}] \tag{27}$$

The dynamic response can then be expressed by the normal mode shapes and modal amplitudes, η_i.

$$x_{i,dyn} = \Phi_i \eta_i \tag{28}$$

The deflection of interior nodes can therefore be expressed as

$$x_i = \Psi_{\mathrm{ib}} x_b + \Phi_i \eta_i \tag{29}$$

Expanding into matrix form, to include boundary and interior nodes

$$\begin{bmatrix} x_b \\ x_i \end{bmatrix} = \begin{bmatrix} x_b \\ \Psi_{\mathrm{ib}} x_b + \Phi_i \eta_i \end{bmatrix} = \begin{bmatrix} I & 0 \\ \Psi_{\mathrm{ib}} & \Phi_i \end{bmatrix} \begin{bmatrix} x_b \\ \eta_i \end{bmatrix} = [R] \begin{bmatrix} x_b \\ \eta_i \end{bmatrix} \tag{30}$$

The matrix $[R]$ is called the reduction basis and reduces the interior degrees of freedom from 6 x No. interior nodes to the number of normal modes *m*.

This response of the original finite element system is expressed as

$$[M] \begin{bmatrix} \ddot{x}_b \\ \ddot{x}_i \end{bmatrix} + [K] \begin{bmatrix} x_b \\ x_i \end{bmatrix} = \begin{bmatrix} f_b \\ f_i \end{bmatrix} \tag{31}$$

The superelement system is defined by substituting in the reduction matrix from (30)

$$[M][R]\begin{bmatrix}\ddot{x}_b\\ \ddot{\eta}_i\end{bmatrix} + [K][R]\begin{bmatrix}x_b\\ \eta_i\end{bmatrix} = \begin{bmatrix}f_b\\ f_i\end{bmatrix} \tag{32}$$

Pre-multiply by $[R]^{\boldsymbol{T}}$

$$[R]^{\boldsymbol{T}}[M][R]\begin{bmatrix}\ddot{x}_b\\ \ddot{\eta}_i\end{bmatrix} + [R]^{\boldsymbol{T}}[K][R]\begin{bmatrix}x_b\\ \eta_i\end{bmatrix} = [R]^{\boldsymbol{T}}\begin{bmatrix}f_b\\ f_i\end{bmatrix} \tag{33}$$

The reduced equations of motion are therefore expressed as follows

$$[\overline{\boldsymbol{M}}]\begin{bmatrix}\ddot{x}_b\\ \ddot{\eta}_i\end{bmatrix} + [\overline{\boldsymbol{K}}]\begin{bmatrix}x_b\\ \eta_i\end{bmatrix} = \overline{\boldsymbol{f}} \tag{34}$$

where

$$[\overline{\boldsymbol{M}}] = [R]^{\boldsymbol{T}}[M][R] \tag{35}$$

$$[\overline{\boldsymbol{K}}] = [R]^{\boldsymbol{T}}[K][R] \tag{36}$$

$$\overline{\boldsymbol{f}} = [R]^{\boldsymbol{T}}\begin{bmatrix}f_b\\ f_i\end{bmatrix} \tag{37}$$

$[\overline{\boldsymbol{M}}]$ is the reduced mass matrix
$[\overline{\boldsymbol{K}}]$ is the reduced stiffness matrix
$\overline{\boldsymbol{f}}$ is the reduced applied load vector