SubDyn ReadMe File - v 1.01.00-rrd

SubDyn is a structural-dynamics module for the analysis of fixed-bottom multi-member substructures. It is based on a linear FEM and a dynamics system reduction and is intended to interface with the FAST modularization framework for offshore wind system modeling.

For running simulations coupled to FAST, please see:
http://wind.nrel.gov/designcodes/simulators/fast8/

Overview

SubDyn is a new module that can be integrated into the FAST modularization framework, thereby allowing for structural-dynamics simulations of wind turbine systems with fixed-bottom multi-member substructures within the FAST aero-hydro-servo-elastic computer-aided engineering (CAE) tool. Substructure types supported by SubDyn include monopiles, tripods, jackets, and other lattice-type structures common for offshore wind installations in shallow and transitional water depths. SubDyn can also be used to model lattice support structures for land-based wind turbines.

SubDyn relies on two main engineering schematizations: (1) a linear frame finite-element beam model (LFEB), and (2) a dynamics system reduction via Craig-Bampton's method (C-B). More details can be found in Song et al (2013), Damiani et al (2013), and Damiani and Song (2013).

In SubDyn, the substructure is considered to be clamped at the seabed and rigidly connected to the transition piece (TP) at the substructure top nodes (interface nodes). Other restraints will be implemented in the future. Only the substructure is intended to be modeled within SubDyn. When integrated with FAST, the structural dynamics of the TP, tower, and rotor/nacelle assembly (RNA) are modeled within FAST's ElastoDyn module (for full lattice support structures or other structures with no transition piece, the entire support structure up to the yaw bearing can be modeled within SubDyn). Loads and responses are transferred between SubDyn, HydroDyn, and ElastoDyn via the FAST driver program (glue code) at each coupling time step. At the interface nodes, the six degree-of-freedom (DOF) TP displacements (three translations and three rotations), velocities, and accelerations are inputs to SubDyn from the ElastoDyn module and the six reaction loads at the transition piece (three forces and three moments) are outputs from SubDyn to the ElastoDyn module. SubDyn also outputs the local substructure displacements, velocities, and accelerations to the HydroDyn module to calculate the local hydrodynamic loads that are inputs for the SubDyn module. In addition, the SubDyn module can calculate member internal reaction loads, as requested by the user. SubDyn can also be driven as a standalone code to compute the mode shapes, natural frequencies, and time-domain responses of substructures uncoupled from FAST in the absence of external loading.

The input file defines the substructure geometry, material properties, constraints, finite-element resolution, number of retained modes in the dynamics system reduction, modal damping coefficients, and auxiliary parameters. The geometry is defined by joint coordinates in the global reference system, with the origin at the intersection of the undeflected tower centerline with mean sea level (MSL). A

member connects two joints; multiple members can use a common joint. Nodes are the resultant of the member refinement into multiple (NDiv input) elements (nodes are located at the ends of each element), and they are calculated by the code. Member properties in this release are outer diameter and wall thickness of tubular segments, and material properties are Young's and shear moduli, and mass density. Member properties are specified at the joints; if properties change from one joint to the other, they will be linearly interpolated for the inner finite elements. Thus a tapered member will be treated as a cylindrical member with step-wise variation of its properties; in a future release, a tapered finite-element formulation will be implemented, and a more accurate representation of tapered member can be obtained.

Major Changes from v0.4

Several improvements and bug fixes have been implemented since version v0.4 and the module has undergone an extensive verification effort with good results. Below are the main changes that the user may notice, but for more information refer to the changelog.txt text file within the official archive.

- Eigensolver bug fixes: the LAPACK solver proved to be unstable in single precision, and now it is solely run in double precision, regardless of the precision used in other parts of the code.
- The input file format has changed. Please refer to the new input file at the end of this document and the following notes:
    - First header line has been removed.
    - Simulation Control Section:
        - SDeltaT: the 'DEFAULT' keyword (in place of 0.0) is now used to indicate that the glue-code time step will be used for time integration by SubDyn.
        - IntMethod: allowed values are now 1-4 (in place of 0-3).
        - SttcSolve: New flag introduced. If set to True, the Static Improvement Method will be used (see the bullet below).
    - FEA and Craig-Bampton Parameters Section:
        - In v0.4, the damping coefficients had to be specified for all retained Craig-Bampton modes, or just once for all the modes (if CBMod=False). In this version, the user can input any number of damping coefficients. In case the number of retained C-B modes (NModes) is larger than the input number of damping coefficients (JDampings), the last damping value will be replicated for all the remainder modes.
    - Base Reaction Joints, Interface Joints, Member, and Member Cosine Matrices Sections:
        - One line with units, below the headers, is expected in all the tables of the input file.
    - Output: Summary & Outfile Section:
        - This section now also contains the parameters previously assigned under the Section titled "Output: Fast/Subdyn Output-File Variables"
- Some of the quantities in the summary file have been fixed. Some of the output matrices were, in fact, being output with wrong values due to an index mismatch. The new summary file is

shorter and does not contain some of the CB method matrices, unless the compiler directive is set.

- Static Improvement Method. This new implementation allows for a reduced number of needed modes to capture the contribution of certain loads (such as static gravity and buoyancy loads or high-frequency loads transferred from the turbine). In the previous version, a large number of internal modes were needed to engage substructural modes excited by static and high-frequency forces. This is no longer needed and fewer modes can be retained while still achieving accurate results (see also Tips and Recommendations). With the Static Improvement Method enabled, all modes that are not considered by the Craig-Bampton reduction are treated quasi-statically.
- There is now the possibility of retaining no internal C-B modes, thus relying solely only on the Static Improvement Method, in those cases where the substructure first eigenfrequencies are much higher than the expected energy-containing modes of the entire system.
- The coupling of SubDyn within FAST now includes full hydro-elastic coupling with the HydroDyn hydrodynamics module.

Known Current Limitations

1. Tight coupling is not yet supported.
2. Limited restraint capabilities are available (only clamped nodes allowed).
3. Only nontapered 2-node Euler-Bernoulli (E-B) or Timoshenko (T) *element formulations* are available (in the future, tapered E-B and tapered Timoshenko will be implemented).
4. Only straight circular members are permitted (in the future, a generic cross-section will be allowed).
5. The number of elements per member (NDiv) is constant throughout the structure.
6. Internal matrices are not stored in sparse form, limiting the total number of possible nodes/DOFs to about 300/1800.
7. Foundation (soil-structure interaction) modeling is not yet available.
8. The dynamics system reduction is performed in the absence of external loading (e.g., hydrodynamic added mass).
9. Gravitational loading does not impact the global substructure stiffness.
10. Loads (gravitational, inertial, hydrodynamic) can only be applied as concentrated loads at element nodes; distributed loads (per unit length) are not yet supported.
11. The overlap of multiple beams connected to a single joint is not modeled with superelements.
12. Member-level outputs are only available for up to 9 nodes of up to 9 members (unless the OutAll flag is enabled).
13. No graphics/animation capability to visualize the substructure geometry, modes, motion, and loads is yet available.
14. Complete documentation of user interaction and theory not yet published. (This document will be made into an official user and theory manual in the future.)

SubDyn Input File Commentary.

Units are in SI system (kg, m, s, N).

Output files will be named Root_Name.SD.EXT, where EXT is the extension belonging to the proper output file (e.g., .SUM for the summary file) and Root_Name is either the root of the FAST input file name, or what is set in the SubDyn Driver Input (variable OutRootName) in case SubDyn is run in standalone mode.

An example input file is given below. No lines should be added or removed from the file, except in tables where the number of rows is specified.

Variable – Commentary

Echo – Select whether or not to have the module output an echo of the input file during model initialization (useful for debugging errors in the input file).

SDdeltaT – Set time step to use for time integration. Set SDdeltaT to "DEFAULT" if the glue code (driver program) time step is desired.

IntMethod – [1-4]:Select one of the following options for the time integrator: 1=RK4 (Runge Kutta 4^{th}-order); 2=AB4 (Adams-Bashforth 4^{th}-order predictor); 3=ABM4 (Adams-Bashforth-Moulton 4^{th}-order predictor-corrector); 4=AM2 (Adams-Moulton implicit 2^{nd}-order).

SttcSolve – If Set to True, the Static Improvement Method will be employed. Through this method, SubDyn will correct the dynamic calculation results with a time-step based static assessment. This allows to reduce the number of retained modes to capture effects such as static gravity and buoyancy loads, and high-frequency loads transferred from the turbine.

FEMMod – Select one of the following options for finite-element formulation: 1=Euler-Bernoulli; 3=Timoshenko. Tapered formulations (2 and 4) not yet implemented.

NDiv – Select number of elements per member. Increasing the number of elements per member may increase accuracy. It is recommended to use NDiv>1 when modeling tapered members.

CBMod – Select True to reduce the number of DOFs via the C-B reduction. If False, then the full finite-element model is retained, Nmodes is ignored, and one JDampings value must be input.

Nmodes – Select number of internal DOFs to retain in the C-B reduction. Nmodes=0 corresponds to a Guyan reduction. Nmodes is ignored if CBMod is set to False.

JDampings – Set value(s) of damping coefficients as percent of critical damping for the retained modes. If the number of JDampings is less than the retained modes, the last value will be replicated for all the remainder modes.

NJoints – Input number of joints.

JointID-JointXss-JointYss-JointZss – Table with NJoints rows to input (X,Y,Z) coordinates, respectively, of member joints in main substructure coordinate system (which is equivalent to the inertial-frame coordinate system in FAST); Z=0 is MSL and positive upward. The first column is the joint identification number (ID, sequential numbering) referred to in other inputs below.

NReact – Input number of joints that will constitute the constraints at the seabed.

RJointID-RctTDXss-RctTDYss-RctTDZss-RctRDXss-RctRDYss-RctRDZss – Table with NReact rows to select joint IDs for the restraint nodes at the seabed. TD and RD refer to translational and rotational DOF restraints in the global coordinate system, respectively. Fixity can only be 1 1 1 1 1 1 (all DOFs are locked) for this release.

NInterf – Input number of joints that will constitute the constraints at the top of the substructure (at the transition piece/TP).

IJointID-ItfTDXss-ItfTDYss-ItfTDZss-ItfRDXss-ItfRDYss-ItfRDZss – Table with NInterf rows to select joint IDs for the restraint nodes at the substructure top (at the TP). TD and RD refer to translational and rotational DOF restraints in the global coordinate system, respectively. Fixity can only be 1 1 1 1 1 1 (all DOFS are locked) for this release.

NMembers – Input number of members in the substructure.

MemberID-MJointID1-MJointID2-MPropSetID1-MPropSetID2-COSMID – Table with NMembers rows. For each member, enter an ID, 1st and 2nd joint ID, and 1st and 2nd property set ID (to be applied at 1st and 2nd joint of the member, e.g., to simulate a tapered beam), respectively; COSMID is ignored in this release. Note: SubDyn expects member joints' z-coordinates to be equal or increasing from joint 1 to joint 2.

NPropSets – Input number of individual material/cross-section (X-section) properties. Each set can be applied to one or both ends of a member or to multiple members. Use two sets to simulate a tapered member, but be sure to increase NDiv>1, as this release will approximate the tapered member as a number of stepped elements.

PropSetID-YoungE-ShearG-MatDens-XsecD-XsecT – Table with NPropSets rows to set property ID number and elastic moduli (E,G), material density, cross-section outer diameter and thickness of circular members, respectively. Only circular members are permitted in this release.

NXPropSets – Set to 0. No other value allowed in this release.

PropSetID-YoungE-ShearG-MatDens-XsecA-XsecAsx-XsecAsy-XsecJxx-XsecJyy-XsecJ0 – Ignored in this release.

NCOSMs – Set to 0. No other value allowed in this release.

COSMID-COSM11-COSM12 -COSM13-COSM21-COSM22-COSM23-COSM31-COSM32-COSM33 – Ignored in this release.

NCmass – Set the number of joints that have concentrated masses/inertias.

CMJointID-JMass-JMXX-JMYY-JMZZ – Table with NCmass rows to set joint ID, concentrated mass, and inertias, respectively, for each concentrated mass and inertial quantities in the global coordinate system.

SSSum – Set to true if a summary file is desired. The summary file contains extensive information on the model and its eigenfrequencies. Additionally, it contains center of mass, equivalent stiffness and mass mass matrix of the substructure. File extension will be ".SD.sum".

OutCOSM – Set to true if cosine matrices of all the members in the undeflected configuration are to be included in the summary file.

OutAll – If set to true, all joint loads will be output in the output file (OutJckF=SubDyn-generated output file, or FAST generated output file).

OutSwtch – Select an option for where to write output data: 1=Root_Name.SD.out (standalone SubDyn output); 2= Included in the FAST output file global to all coupled modules; 3=both files.

TabDelim – Set to True if tab-delimited output file is desired.

OutDec – Input rate of decimation for output (10 = every 10^{th} time step is written to the OutJckF file).

OutFmt – Fortran format for the numerical output in the OutJckF file.

OuStFmt – Fortran format for the header strings in the OutJckF file.

NMOutputs – Input number of members for which specific output is requested per the following lines (must be less than or equal to 9).

MemberID-NOutCnt-NodeCnt – Table with NMOutputs rows to set where output is available for each member: input member ID, how many nodes are queried in that member, and the sequence of nodes. For instance, if NDiv=4 there are 5 nodes per member, so, NOutCnt=3, NodeCnt=2 4 5 means that the 2^{nd}, 4^{th}, and 5^{th} nodes of the member are queried (count is from 1^{st} node of the member as defined in member table above). The quantities requested at these nodes are input in the next section of the SubDyn input file.

SSOutList – Select desired output variables, with strings such as M2N3FKze, which requests the elastic force (FK) along the local z-axis of member 2 (M2) at the third node (N3) in the list of NodeCnt (node ID=5 in the above example). Note: some of output variables are tied to the selected queried members and nodes as described above; others are global, e.g., ReactZss, which would return the overall reaction at the seabed at the structure centerline. A complete list of outputs can be found in the Excel spreadsheet named SubDynOutListParameters.xlsx.

Modeling Guidelines

SubDyn allows for the specification of arbitrary multi-member structure geometries. The user defines the geometry by providing a list of joints that represent the endpoints along a beam; one or more members are connected at each joint. Members and their cross-sectional properties are then defined

between two joints. Members can be further subdivided into multiple (NDiv) sub-elements to increase the model resolution; nodes, where the numerical calculations take place, are located at the endpoints of each element. In order to keep the mesh as uniform as possible when using NDiv, the initial member definition should also have rendered a uniform mesh. For tapered members, it is recommended to set NDiv>1.

Improper discretization of the members may decrease the accuracy of the model.

When SubDyn is coupled to FAST, the joints and members need not match between HydroDyn and SubDyn—FAST's mesh-mapping utility handles transfer of motion and loads across meshes in a physically relevant manner [Sprague et al (2014)], but consistency between the joints and members in HydroDyn and SubDyn is advised. In ElastoDyn, the 6 DOFs of the platform must be enabled in order to couple loads and displacements between turbine and substructure. The platform reference point coordinates in ElastoDyn should also be set equal to the TP reference point (normally the tower base flange location or TP center). A rigid connection between SubDyn interface joints and TP reference point (≡ platform reference point) will be assumed. ElastoDyn also needs tower mode shapes specified (coefficients of best fit 6th order polynomials). They can be derived with an appropriate software (finite element analysis, energy methods, or analytically) and making use of the SubDyn-derived equivalent substructure stiffness and mass matrices ($\tilde{K}_{BB}$ and $\tilde{M}_{BB}$ matrices found in the SubDyn summary file) for the boundary conditions at the base of the tower. For instance, using NREL's BModes software, the SubDyn-obtained matrices can be used in place of the hydrodynamic stiffness (hydro_K) and mass matrices (hydro_M); by setting the hub_conn boundary condition to 2 (free-free), BModes will calculate the mode shapes of the tower when tower cross-sectional properties are supplied. In order to obtain eigenmodes that are compatible with the FAST modal treatment of the tower (i.e., no axial or torsional modes, and no distributed rotational-inertia contribution to the eigenmodes) the tower distributed properties should be modified accordingly (e.g., by reducing mass moments of inertia towards zero, and by increasing torsional and axial stiffness, while assuring convergence of the results; see also https://wind.nrel.gov/forum/wind/viewtopic.php?f=4&t=742).

Furthermore, it is recommended that the ratio of element length between elements of HydroDyn and SubDyn not exceed 10 to 1. Due to the exponential decay of hydrodynamic loads with depth, HydroDyn requires higher resolution near the water free surface to capture hydrodynamic loading as waves oscillate about the still water level (SWL). It is recommended, for instance, that the HydroDyn discretization not exceed element lengths of 0.5 m in the region of the free surface (5- to 10 m above and below SWL), 1.0 m down to 25- to 50-m depth, and 2.0 m in deeper waters. As such, it is recommended that the SubDyn discretization not exceed element lengths of 5 m in the region of the free surface, 10 m down to 25- to 50-m depth, and 20 m in deeper waters. These are not absolute rules, but rather a good starting point that will likely require refinement for a given substructure. Additional considerations for SubDyn discretization include aspects that will impact structural accuracy such as member weight, substructure modes/natural frequencies, load transfer, tapered members, etc.

Following the above guidelines will lead to large numbers of joints and members, resulting in high DOF simulations. This makes the C-B reduction useful to reduce computational expense. In SubDyn, the user

needs to identify the number of C-B modes to retain (retaining all modes results in a model effectively equivalent to the full FEM formulation and retaining no modes results in the Guyan reduction). However, the minimum number of modes to retain is specific to each substructure design and application, and will change if the total number of DOFs is changed by altering the discretization.

When SubDyn is coupled with FAST, it is important to choose the number of C-B modes in SubDyn that ensure that the physical modes of the coupled system are properly captured by the coupled model. It is recommended that all physical modes up to at least 2-3 Hz be captured; wind, wave and turbine excitations are important for frequencies up to 2-3 Hz. Eigen analysis of the linearized, coupled system will make checking this condition possible; however the linearization process has yet to be implemented in FAST v8. Before then, experience has shown that it is sufficient to enable all C-B modes up to 10 Hz. (The natural frequencies of the C-B modes are written to the SubDyn summary file.) If the Static Improvement Method is not enabled, in addition to capturing physical modes up to a given frequency, the highest C-B mode must include the substructure axial mode so that gravity loading from self-weight is properly accounted for within SubDyn. This likely requires enabling a high number of C-B modes, reducing the benefit of the C-B reduction. Thus, it is recommended employing the C-B reduction with the Static Improvement Method enabled. Due to the fixed-fixed treatment of the substructure boundary conditions in the C-B reduction, the C-B modes will always have higher natural frequencies than the physical modes.

Another consideration for creating SubDyn input files is the time step size. SubDyn offers three explicit time-integrators—the fourth-order Runge-Kutta (RK4), fourth-order Adams-Bashforth (AB4), fourth-order Adams-Bashforth-Moulton (ABM4) methods—and the implicit second-order Adams-Moulton (AM2) method. The user has the option to use the global time step from the glue code or an alternative SubDyn-unique time step that is an integer multiple smaller than the glue code time step. It is essential that a small enough time step is used to ensure solution accuracy (by providing a sufficient sampling rate to characterize all key frequencies of the system), to ensure numerical stability of the selected explicit time-integrator, and to ensure that the coupling with FAST is numerically stable. For the RK4 and ABM4 methods, it is recommended that the SubDyn time step follow the relationship shown in Eq. (1), where $f_{\max}$ is the higher of the frequencies in Hz of (1) the highest natural frequency of the retained C-B modes and (2) the highest natural frequency of the physical modes when coupled to FAST. While the former can be obtained from the SubDyn summary file, the latter is hard to estimate before the full-system linearization of the coupled FAST model is realized. Until then, experience has shown that the highest physical mode when SubDyn is coupled to FAST is often the platform-heave mode of ElastoDyn, with a frequency given by Eq. (2), where $K_{33}^{(SD)}$ is the equivalent heave stiffness of the substructure (the (3,3) element of the $\tilde{K}_{BB}$ matrix found in the SubDyn summary file), $M_{33}^{(SD)}$ is the equivalent heave mass of the substructure (the (3,3) element of the $\tilde{M}_{BB}$ matrix found in the SubDyn summary file), and $M^{(ED)}$ is the total mass of the rotor, nacelle, tower, and TP (found in the ElastoDyn summary file).

$$dt_{\max} = \frac{1}{10\left(f_{\max}\right)} \tag{1}$$

$$f = \frac{1}{2\pi}\sqrt{\frac{K_{33}^{(SD)}}{(M_{33}^{(SD)} + M^{(ED)})}} \tag{2}$$

For the AB4 method, the recommend time step is half the value given by Eq. (1). For AM2, being implicit, the required time step is not driven by natural frequencies within SubDyn, but should still be chosen to ensure solution accuracy and that the coupling to FAST is numerically stable.

The C-B method assumes no damping for the interface modes. This is equivalent to having six undamped rigid-body DOFs at the TP reference point in the absence of aerodynamic or hydrodynamic damping. Experience has shown that negligible platform-heave damping can cause numerical problems when SubDyn is coupled to FAST. One way to overcome this problem is to augment other dampings in the system with an additional linear damping for the platform-heave DOF. This can be achieved quite easily by calculating the damping from Eq. (3) and specifying this as the (3,3) element of HydroDyn's additional linear damping matrix, *AddBLin*. Experience has shown that a damping ratio of 1% of critical ($\zeta = 0.01$) is sufficient.

$$C_{33}^{(HD)} = 2\zeta\sqrt{K_{33}^{(SD)}\left(M_{33}^{(SD)} + M^{(ED)}\right)} \tag{3}$$

To minimize extraneous excitation of the platform-heave DOF, it is useful to set the initial platform-heave displacement to its natural static-equilibrium position, which can be approximated by Eq. (4), where g is the magnitude of gravity. $PtfmHeave$ from Eq. (4) should be specified in the initial condiditions section of the ElastoDyn input file.

$$PtfmHeave = -\frac{(M_{33}^{(SD)} + M^{(ED)})g}{K_{33}^{(SD)}} \tag{4}$$

References:

Damiani, R.; Song, H.(2013) Jacket Sizing Tool for Offshore Wind Turbines Within the Systems Engineering Initiative. ; NREL Report No. CP-5000-57492.

Damiani, R.; Jonkman, J.; Robertson, A.; Song, H. (2013). Assessing the Importance of Nonlinearities in the Development of a Substructure Model for the Wind Turbine CAE Tool FAST: Preprint. 18 pp.; NREL Report No. CP-5000-57850.

Song, H.; Damiani, R.; Robertson, A.; Jonkman, J. (2013). New Structural-Dynamics Module for Offshore Multimember Substructures within the Wind Turbine Computer-Aided Engineering Tool FAST: Preprint. 12 pp.; NREL Report No. CP-5000-58093.

Sprague, M. A.; Jonkman, J. M.; Jonkman, B. J. (2014). FAST Modular Wind Turbine CAE Tool: Nonmatching Spatial and Temporal Meshes. Proceedings of the 32nd ASME Wind Energy Symposium, 13-17 January 2014, National Harbor, Maryland. New York, NY: American Society of Mechanical Engineers (ASME) 24 pp.; NREL Report No. CP-2C00-61839.

SAMPLE INPUT FILE

```
---------- SubDyn v1.01.x MultiMember Support Structure Input File ------------
OC3 Monopile configuration (pile only).
------------------------- SIMULATION CONTROL  ---------------------------------
False            Echo        - Echo input data to "<rootname>.SD.ech" (flag)
DEFAULT          SDdeltaT    - Local Integration Step. If "default", the glue-code integration step will be used.
             3   IntMethod   - Integration Method [1/2/3/4 = RK4/AB4/ABM4/AM2].
True             SttcSolve   - Solve dynamics about static equilibrium point
-------------------- FEA and CRAIG-BAMPTON PARAMETERS----------------------------
             3   FEMMod      - FEM switch: element model in the FEM. [1= Euler-Bernoulli(E-B) ; 2=Tapered E-B (unavailable); 3= 2-node Timoshenko;  4= 2-node tapered
Timoshenko (unavailable)]
             3   NDiv        - Number of sub-elements per member
True             CBMod       - [T/F] If True perform C-B reduction, else full FEM dofs will be retained. If True, select Nmodes to retain in C-B reduced system.
             0   Nmodes      - Number of internal modes to retain (ignored if CBMod=False). If Nmodes=0 --> Guyan Reduction.
           1.0   JDampings   - Damping Ratios for each retained mode (% of critical) If Nmodes>0, list Nmodes structural damping ratios for each retained mode (% of
critical), or a single damping ratio to be applied to all retained modes. (last entered value will be used for all remaining modes).
---- STRUCTURE JOINTS: joints connect structure members (~Hydrodyn Input File)---
             4   NJoints     - Number of joints (-)
JointID          JointXss               JointYss               JointZss  [Coordinates of Member joints in SS-Coordinate System]
  (-)              (m)                    (m)                    (m)
   1              0.00000                0.00000              -20.00010
   2              0.00000                0.00000              -10.00000
   3              0.00000                0.00000                0.00000
   4              0.00000                0.00000               10.00000
------------------- BASE REACTION JOINTS: 1/0 for Locked/Free DOF @ each Reaction Node ----------------------
             1   NReact      - Number of Joints with reaction forces; be sure to remove all rigid motion DOFs of the structure  (else det([K])=[0])
RJointID   RctTDXss    RctTDYss    RctTDZss    RctRDXss    RctRDYss    RctRDZss     [Global Coordinate System]
  (-)       (flag)      (flag)      (flag)      (flag)      (flag)      (flag)
   1           1           1           1           1           1           1
------- INTERFACE JOINTS: 1/0 for Locked (to the TP)/Free DOF @each Interface Joint (only Locked-to-TP implemented thus far (=rigid TP)) ---------
             1   NInterf     - Number of interface joints locked to the Transition Piece (TP):  be sure to remove all rigid motion dofs
IJointID   ItfTDXss    ItfTDYss    ItfTDZss    ItfRDXss    ItfRDYss    ItfRDZss     [Global Coordinate System]
  (-)       (flag)      (flag)      (flag)      (flag)      (flag)      (flag)
   4           1           1           1           1           1           1
---------------------------------- MEMBERS ------------------------------------
             3   NMembers    - Number of frame members
MemberID   MJointID1   MJointID2   MPropSetID1   MPropSetID2     COSMID
  (-)         (-)         (-)          (-)           (-)           (-)
   1           1           2           1             1
   2           2           3           1             1
   3           3           4           1             1
------------------ MEMBER X-SECTION PROPERTY data 1/2 [isotropic material for now: use this table for circular-tubular elements] ------------------------
             4   NPropSets   - Number of structurally unique x-sections (i.e. how many groups of X-sectional properties are utilized throughout all of the members)
PropSetID     YoungE          ShearG          MatDens          XsecD           XsecT
  (-)         (N/m2)          (N/m2)          (kg/m3)            (m)             (m)
   1        2.10000e+11     8.08000e+10       8500.00         6.000000        0.060000
   2        2.10000e+11     8.08000e+10       8500.00         6.000000        0.027000
   3        2.10000e+11     8.08000e+10       8500.00         3.870000        0.019000
   4        2.10000e+14     8.08000e+14          1.00         1.000000        0.010000
------------------ MEMBER X-SECTION PROPERTY data 2/2 [isotropic material for now: use this table if any section other than circular, however provide COSM(i,j) below] ----
--------------------
             0   NXPropSets  - Number of structurally unique non-circular x-sections (if 0 the following table is ignored)
PropSetID     YoungE          ShearG          MatDens          XsecA          XsecAsx       XsecAsy       XsecJxx       XsecJyy        XsecJ0
  (-)         (N/m2)          (N/m2)          (kg/m3)          (m2)             (m2)          (m2)          (m4)          (m4)          (m4)
---------------------- MEMBER COSINE MATRICES COSM(i,j) ------------------------
             0   NCOSMs      - Number of unique cosine matrices (i.e., of unique member alignments including principal axis rotations); ignored if NXPropSets=0   or 9999
in any element below
COSMID    COSM11    COSM12    COSM13    COSM21    COSM22    COSM23    COSM31    COSM32    COSM33
 (-)       (-)       (-)       (-)       (-)       (-)       (-)       (-)       (-)       (-)
------------------------ JOINT ADDITIONAL CONCENTRATED MASSES--------------------------
             0   NCmass      - Number of joints with concentrated masses; Global Coordinate System
CMJointID       JMass            JMXX             JMYY             JMZZ
  (-)             (kg)          (kg*m^2)         (kg*m^2)         (kg*m^2)
---------------------------- OUTPUT: SUMMARY & OUTFILE ------------------------------
```

```
True             SSSum       - Output a Summary File (flag).It contains: matrices K,M  and C-B reduced M_BB, M-BM, K_BB, K_MM(OMG^2), PHI_R, PHI_L. IT can also contain
COSMs if requested.
False            OutCOSM     - Output cosine matrices with the selected output member forces (flag)
False            OutAll      - [T/F] Output all members' end forces in OutJckF (flag)
             2   OutSwtch    - [1/2/3] Output requested channels to: 1=<rootname>.SD.out (this is OutJckF); 2=<rootname>.out (generated by FAST);  3=both files.
True             TabDelim    - Generate a tab-delimited output file in OutJckF
             1   OutDec      - Decimation of output for OutJckF only
"ES11.4e2"       OutFmt      - Output format for numerical results in OutJckF
"A11"            OutSFmt     - Output format for header strings
------------------------ MEMBER OUTPUT LIST --------------------------------------
             2   NMOutputs   - Number of members whose forces/displacements/velocities/accelerations will be output into OutJckF (-) [Must be <= 9].
MemberID   NOutCnt    NodeCnt [NOutCnt=how many nodes to get output for [< 10]; NodeCnt are local ordinal numbers from the start of the member, and must be >=1 and <=
NDiv+1] If NMOutputs=0 leave blank as well.
  (-)        (-)        (-)
   2          1        1.000
   3          1        1.000
------------------------ SSOutList: The next line(s) contains a list of output parameters that will be output in <rootname>.SD.out or <rootname>.out. ------
"M2N1MKxe, M2N1MKye"                - The local side-to-side and fore-aft bending moments at node 1 of member 3 (located at 0 m, i.e. MSL).
"M1N1MKxe, M1N1MKye"                - The local side-to-side and fore-aft bending moments at node 1 of member 2 (located at -10 m, i.e. half way between MSL and mudline).
"-ReactFXss, -ReactFYss, -ReactFZss" - Base reactions: fore-aft shear, side-to-side shear and vertical forces at the mudline.
"-ReactMXss, -ReactMYss, -ReactMZss" - Base reactions: side-to-side, fore-aft and yaw moments at the mudline.
END of output channels and end of file. (the word "END" must appear in the first 3 columns of this line)
```