

## SubDyn ReadMe File

*SubDyn is a structural dynamics code for the analysis of offshore substructures for horizontal-axis wind turbines. It is based on a linear FEM and a dynamics system reduction and is intended to interface with the FAST modularization framework.*

For running simulations coupled to FAST, please see:

<http://wind.nrel.gov/designcodes/simulators/fast/alpha/>

## 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 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. 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 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 transition piece, tower, and rotor/nacelle assembly 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 transition piece displacement, rotation, velocity, and acceleration are inputs from the ElastoDyn module and the reaction forces at the transition piece are outputs of the SubDyn module. SubDyn also outputs the substructure displacements, velocities, and accelerations to the HydroDyn module to calculate the hydrodynamic loads that are inputs for the SubDyn module. The SubDyn module can calculate the member loads, as requested by the user. In addition, the SubDyn module can calculate the member 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, tapered finite-element formulation will be implemented, and a more accurate representation of tapered member can be obtained.

As of September 2013, the module has been released in alpha form as a FORTRAN code (v0.04.00a) with only limited application to realistic cases (see Section below). Hydroelastic coupling of SubDyn and HydroDyn has not yet been implemented. Continuous work is ongoing to verify the code against other tools and to expand its capabilities.

*Please use caution in interpreting the results and check back often to obtain the latest version of the code.*

## Known Current Limitations

1. Tight coupling is not yet supported.
2. Only limited error checking of user's input is implemented. Carefully examine the input file if unexpected errors are found.
3. Limited restraint capabilities are available (only clamped nodes allowed).
4. Only nontapered Euler-Bernoulli (E-B) or a 2-node Timoshenko (T) element formulations are available (in the future, tapered E-B and tapered T will be implemented).
5. Only circular members are permitted (in the future, a generic cross-section will be allowed).
6. Reaction loads are calculated at (0,0,-Wdepth) in the substructure (SS) reference frame (e.g., sea-bed level); in the future, a reference location will be user input.
7. The number of elements per member (NDiv) is constant throughout the structure.
8. The code has not yet been coupled with HydroDyn.
9. Limited testing has been conducted; use caution in interpreting the results and try different numbers of retained modes to ensure convergence of the results if using CBMod.
10. Internal matrices are not stored in sparse form, limiting the total number of possible nodes/degrees of freedom (DOF) to about 300/1800.
11. Foundation (soil-structure interaction) modeling is not yet available.
12. The dynamics system reduction is performed in the absence of external loading.
13. Gravitational loading does not impact the global substructure stiffness.
14. Loads (gravitational, inertial, hydrodynamic) can only be applied as concentrated loads at element nodes; distributed loads (per unit length) are not yet supported.
15. Compiling requires the Intel Math Kernel Library (gfortran is not yet supported). Note that IMKL routines used in this release are publicly available LAPACK routines at <http://www.netlib.org/lapack>.

## SubDyn Input File Commentary.

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

Output files will be named Root\_Name.EXT, where EXT is the extension belonging to the proper output file (e.g., .SUM for the summary file) and Root\_Name is the root of the input file name.

An example input file is given below. No lines should be added or removed from the file.

### Variable - Commentary

Echo – Select whether or not to have the code output an echo of the input file during model initialization.

SDdeltaT – Set time step to use for integration. Set SDdeltaT to 0.0 if the glue code (driver program) time step is desired.

IntMethod – Select one of the following options for the time integrator: RK4(Runge Kutta 4<sup>th</sup> order);AB4(Adams Bashforth 4<sup>th</sup> order);/(ABM4 (Adams-Bashforth-Moulton Predictor-Corrector Method 4<sup>th</sup> order). Tight Coupling not yet implemented.

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 may increase accuracy. It is recommended to use NDiv>1 when using tapered members.

CBMod – Select True to reduce the number of DOFs via the Craig-Bampton 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. *Be careful: a sufficient number of DOFs must be retained in order to excite modes that would transfer loads throughout the structure.* For instance, in order for gravity to take effect, the number of modes to retain should be large enough to retain at least the first extensional (vertical) mode of the structure. Nmodes is ignored if CBMod is set to False.

JDampings – Set values of damping coefficients as percent of critical damping for the retained modes. If CBMod=False, set only one value that will be used for all modes.

NJoints – Input number of joints.

JointID-JointXss-JointYss-JointZss – Input (X,Y,Z) coordinates of member joints in main substructure coordinate system; Z=0 is 0 MSL. 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.

JointID,RctTDXss-RctTDYss-RctTDZss –RctRDXss-RctRDYss-RctRDZss – 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. Fixity can only be 111111 (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).

JointID-ItfTDXss-ItfTDYss-ItfTDZss-ItfRDXss-ItfRDYss-ItfRDZss – 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. Fixity can only be 111111 (all DOFS are locked) for this release.

NMembers – Input number of members in the substructure

MemberID-MJointID1-MJointID2-MPropSetID1-MPropSetID2-COSMID – For each member, enter an ID, 1<sup>st</sup> and 2<sup>nd</sup> joint ID, and 1<sup>st</sup> and 2<sup>nd</sup> property set ID (to be applied at 1<sup>st</sup> and 2<sup>nd</sup> joint of the member, e.g., to simulate a tapered beam); COSMID ignored in this release.

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 – Set property ID number and elastic moduli (E,G), material density, cross-section outer diameter and thickness. 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 number of joints that have concentrated masses.

JointID-JMass-JMXX-JMYJ- JMZZ – Set joint ID 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 rigid body mass matrix of the substructure. File extension will be “.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 forces will be output in the .OUT file.

OutSwitch – Select an option for where to write output data: 1= Root\_Name.out, 2=GlueCode.out, 3=both files.

NMOutputs – Input number of members for which specific output is requested per the following lines.

MemberID-NOutCnt-NodeCnt – For each member for which output is requested: input member ID, how many nodes are queried in that member, and the sequence of Nodes, (for instance NDiv=4, ➔ 5 nodes per member, NOutCnt=3, NodeCnt= 2 4 5 ➔ 2<sup>nd</sup>, 4<sup>th</sup>, and 5<sup>th</sup> nodes of the member are queried (count is from 1<sup>st</sup> node of the member as defined in previous lines). The quantities requested at these nodes are input in the following section of the input file, with strings such as M2N5FKz, which requests the elastic force (FK) along the local z-axis of member 2 (M2) at the fifth node (N5).

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

OutDec – Input rate of decimation for output (10 = every 10<sup>th</sup> time step is written to the file).

OutFmt – Format for the numerical output in the .OUT file.

OutStFmt – Format for the header strings in the .OUT file.

SSOutList – Select desired output variables. Note: some of them 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.

## 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.

```

----- SubDyn MultiMember Support Structure Input File -----
Example of Jacket Structure Input File. Note >5 modes needed to capture gravitational effects!
----- SIMULATION CONTROL -----
True      Echo      - Echo input data to "echo.out" (flag)
0.0       SDDeltaT   - Local Integration Step (s). If 0.0, the glue-code integration step will be used.
1         IntMethod  - Integration Method (0/1/2/3 = GlueCode(tight Coupling)/RK4/AB4/ABM4)
----- FEA and CRAIG-BAMPTON PARAMETERS -----
1         FEMMod     - FEM switch (-): element model in the FEM:1=Euler-Bernoulli(E-B);2=Tapered E-B;3=2-node Timoshenko;4=2-node tapered Timoshenko
1         NDiv       - Number of sub-elements per member (-)
true      CBMod      - CB_flag. If True, perform C-B reduction, else full FEM dofs will be retained. IF True, select modes to retain in C-B
12        Nmodes     - Number of internal modes to retain (ignored if CBMod=False) (-). If Nmodes=0 --> Guyan Reduction.
2 2 2 2 2 2 2 2 2 2 JDampings - Damping Ratios for each retained mode (% of critical)
---- STRUCTURE JOINTS: joints connect structure members (~Hydrodyn Input File)---
5         NJoints    - Number of joints (-)
JointID   JointXss   JointYss   JointZss   -Coordinates of Member joints in SS-Coordinate System
(-)       (m)        (m)          (m)
1         -10.6066   -10.6066   0.0
2         -10.6066   -10.6066   20.0
3         0.0        0.0        40.0
4         10.6066    10.6066    20.0
5         10.6066    10.6066    0.0
----- BASE REACTION JOINTS: 1/0 for Locked/Free DOF @ each Reaction Node -----
2         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
1         1         1         1         1         1         1
5         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
3         1         1         1         1         1         1
----- MEMBERS -----
5         NMembers   - Number of frame members (-)
MemberID  MJointID1  MJointID2  MPropSetID1 MPropSetID2 COSMID
1         1         2         1         1
2         5         4         1         1
3         2         4         1         1
4         2         3         2         2
5         3         4         2         2
----- MEMBER X-SECTION PROPERTY data 1/2 [isotropic material for now: use this table if circular-tubular elements -----
2         NPropSets  - Number of structurally unique x-sections (-) (i.e., how many groups of X-sectional properties are utilized throughout all members)
           YoungE     ShearG       MatDens       XsecD       XsecT
PropSetID (N/m2)      (N/m2)      (kg/m3)      (m)        (m)
1         2.10E+11    8.0769E+10  7850.0      3.0        0.1
2         2.10E+11    8.0769E+10  7850.0      2.0        0.2
-----MEMBER X-SECTION PROPERTY data 2/2 (isotropic material for now: use table if any section other than circular, and provide COSM(i,j) below) ----
0         NXPropSets - Number of structurally unique non-circular x-sections (if 0 the following table is ignored)
           YoungE     ShearG       MatDens       XsecA       XsecAsx     XsecAsy     XsecJxx     XsecJyy     XsecJO

```

```

PropSetID      (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
COSMID      COSM11      COSM12      COSM13      COSM21      COSM22      COSM23      COSM31      COSM32      COSM33
----- JOINT ADDITIONAL CONCENTRATED MASSES -----
1      NCmass      - Number of joints with concentrated masses (-); Global Coordinate System
CMJointID      JMass      JMX      JMY      JMZZ
              (kg)      (kg*m2)      (kg*m2)      (kg*m2)
3      3.5000e+05      0.0      0.0      0.0
----- OUTPUT: SUMMARY & OUTFILE -----
True      SSSum      - Output Summary File (flag).It contains: matrices K, M; C-B reduced M_BB, M-BM, K_BB, K_MM(OMG^2), PHI_R, PHI_M;COSMs if
requested
True      OutCOSM      - Output cosine matrices with the selected output member forces (flag)
True      OutAll      - [T/F] Output all members' end forces
1      OutSwch      - [1/2/3] Output requested channels to: 1=(SubDyn RootName).out 2=GlueCode.out 3=both files
----- MEMBER OUTPUT LIST -----
2      NMOoutputs      - Number of members whose forces/displacements/velocities/accelerations will be output (-) [Must be <= 9]
MemberID      NOutCnt      NodeCnt [NOutCnt=how many nodes to get output for [<10];NodeCnt=local ordinal numbers from the member start joint, must be >=1
and <= NDiv+1] If NMOoutputs=0 leave blank
(-)      (-)      (-)
1      2      1 2
3      2      1 2
----- OUTPUT: FAST/SUBDYN OUTPUT-FILE VARIABLES -----
True      TabDelim      - Generate a tab-delimited output file (flag)
10      OutDec      - Decimation of output (-)
"ES11.4e2"      OutFmt      - Output format for numerical results
"All"      OutSFmt      - Output format for header strings (quoted string) [not checked for validity!]
----- SSOutList - The next line(s) contains a list of output parameters that will be output in the main FAST .out. -----
"ReactXss, ReactYss, ReactZss, ReactMXss, ReactMYss, ReactMZss" -Base reactions (forces onto SS structure)
"IntfXss, IntfYss, IntfZss, IntfMXss, IntfMYss, IntfMZss" -Interface reactions (forces from SS structure)
"IntfTDXss, IntfTDYss, IntfTDZss, IntfRDXss, IntfRDYss, IntfRDZss" -Interface deflections
"IntfTAXss, IntfTAYss, IntfTAZss, IntfRAXss, IntfRAYss, IntfRAZss" -Interface accelerations
"M1N1FKze ,M1N1MKxe ,M1N1MKye " - 1st Member, 1st node cnt, static components of axial force & and bending moments
"SSqm01,SSqm02,SSqm03" - values of generalized modal variable 1,2,7
"SSqmd01, SSqmd02, SSqmd03" - values of 1st time derivative of generalized modal variables 1,2,7
"SSqmd01,SSqmd02,SSqmd03" - values of 2nd time derivative of generalized modal variables 1,2,7
END of SubDyn input file (the word "END" must appear in the first 3 columns of this last line).
-----

"M1N1FMze ,M1N1MMxe ,M1N1MMye " - 1st Member, 1st node cnt, dynamic components of axial force and bending moments
"M2N4FKze ,M2N4MKxe ,M2N4MKye " - 2nd Member, 4th node cnt , static components of axial force and bending moments
"M2N4FMze ,M2N4MMxe ,M2N4MMye " - 2nd Member, 4th node cnt, dynamic components of axial force and bending moments
"M3N3TDxss,M3N3TDyss,M3N3TDzss,M3N3RDxe,M3N3RDye,M3N3RDze" - 3rd Member, 3rd node, x,y,z linear and rotational displacements
"M4N3TAxe,M4N3TAye,M4N3TAze,M4N3RAXe,M4N3RAYe,M4N3RAze" - 4th Member, 3rd node, x,y,z linear and rotational accelerations

```