4.2.1.3. Input Files

The user configures the aerodynamic model parameters via a primary AeroDyn input file, as well as separate input files for airfoil and blade data. When used in standalone mode, an additional driver input file is required. The AeroDyn driver and driver input file are detailed in Section 4.2.1.6. The driver file specifies initialization inputs normally provided to AeroDyn by OpenFAST, as well as the per-time-step inputs to AeroDyn.

As an example, the `driver.dvr` file is the main driver, the `input.dat` is the primary input file, the `blade.dat` file contains the blade geometry data, and the `airfoil.dat` file contains the airfoil angle of attack, lift, drag, moment coefficients, and pressure coefficients. Example input files are included in Section 4.2.1.11.

No lines should be added or removed from the input files, except in tables where the number of rows is specified and comment lines in the AeroDyn airfoil data files.

4.2.1.3.1. Units

AeroDyn uses the SI system (kg, m, s, N). Angles are assumed to be in radians unless otherwise specified.

4.2.1.3.2. AeroDyn Primary Input File

The primary AeroDyn input file defines modeling options, environmental conditions (except freestream flow), airfoils, tower nodal discretization and properties, tower, hub, and nacelle buoyancy properties, as well as output file specifications.

The file is organized into several functional sections. Each section corresponds to an aspect of the aerodynamics model. A sample AeroDyn primary input file is given in Section 4.2.1.11.

The input file begins with two lines of header information which is for your use, but is not used by the software.

4.2.1.3.2.1. General Options

Set the `Echo` flag to TRUE if you wish to have AeroDyn echo the contents of the AeroDyn primary, airfoil, and blade input files (useful for debugging errors in the input files). The echo file has the naming convention of *OutRootFile.AD.ech*. `OutRootFile` is either specified in the I/O SETTINGS section of the driver input file when running AeroDyn standalone, or by the OpenFAST program when running a coupled simulation.

`DTAero` sets the time step for the aerodynamic calculations. For accuracy and numerical stability, we recommend that `DTAero` be set such that there are at least 200 azimuth steps per rotor revolution. However, when AeroDyn is coupled to OpenFAST, OpenFAST may require time steps much smaller than this rule of thumb. If UA is enabled while using very small time steps, you may need to recompile AeroDyn in double precision to avoid numerical problems in the UA routines. The keyword `DEFAULT` for `DTAero` may be used to indicate that AeroDyn should employ the time step prescribed by the driver code (OpenFAST or the standalone driver program).

Set `WakeMod` to 0 if you want to disable rotor wake/induction effects or 1 to include these effects using the (quasi-steady) BEM theory model. When `WakeMod` is set to 2, a dynamic BEM theory model (DBEMT) is used (also referred to as dynamic inflow or dynamic wake model, see Section 4.2.1.7.2). When `WakeMod` is set to 3, the free vortex wake model is used, also referred to as OLAF (see Section 4.2.2). `WakeMod` cannot be set to 2 or 3 during linearization analyses.

Set `AFAeroMod` to 1 to include steady blade airfoil aerodynamics or 2 to enable UA; `AFAeroMod` must be 1 during linearization analyses with AeroDyn coupled to OpenFAST.

Set `TwrPotent` to 0 to disable the potential-flow influence of the tower on the fluid flow local to the blade, 1 to enable the standard potential-flow model, or 2 to include the Bak correction in the potential-flow model.

Set the `TwrShadow` to 0 to disable to the tower shadow model, 1 to enable the Powles tower shadow model, or 2 to use the Eames tower shadow model. These models calculate the influence of the tower on the flow local to the blade based on the downstream tower shadow model. If the tower influence from potential flow and tower shadow are both enabled, the two influences will be superimposed.

Set the `TwrAero` flag to TRUE to calculate fluid drag loads on the tower or FALSE to disable these effects.

During linearization analyses with AeroDyn coupled OpenFAST and BEM enabled (`WakeMod = 1`), set the `FrozenWake` flag to TRUE to employ frozen-wake assumptions during linearization (i.e. to fix the axial and tangential induces velocities, and, at their operating-point values during linearization) or FALSE to recalculate the induction during linearization using BEM theory.

Set the `CavitCheck` flag to TRUE to perform a cavitation check for MHK turbines or FALSE to disable this calculation. If `CavitCheck` is TRUE, `AFAeroMod` must be set to 1 because the cavitation check does not function with unsteady airfoil aerodynamics. If `CavitCheck` is TRUE, the `MHK` flag in the AeroDyn or OpenFAST driver input file must be set to 1 or 2 to indicate an MHK turbine is being modeled.

Set the `Buoyancy` flag to TRUE to calculate buoyant loads on the blades, tower, nacelle, and hub of an MHK turbine or FALSE to disable this calculation. If `Buoyancy` is TRUE, the `MHK` flag in the AeroDyn or OpenFAST driver input file must be set to 1 or 2 to indicate an MHK turbine is being modeled.

Set the `CompAA` flag to TRUE to run aero-acoustic calculations. This option is only available for `WakeMod = 1` or `2` and is not available for an MHK turbine. See section Section 4.2.3 for information on how to use this feature.

The `AA_InputFile` is used to specify the input file for the aeroacoustics sub-module. See Section 4.2.3 for information on how to use this feature.

4.2.1.3.2.2. Environmental Conditions

Environmental conditions are now specified in driver input files but are left in the AeroDyn primary input file for legacy compatibility. Use the keyword `DEFAULT` to pass in values specified by the driver input file. Otherwise, values given in the AeroDyn primary input file will overwrite those given in the driver input file. `AirDens` specifies the fluid density and must be a value greater than zero; a typical value is around 1.225 kg/m^3 for air (wind turbines) and 1025 kg/m^3 for seawater (MHK turbines). `KinVisc` specifies the kinematic viscosity of the fluid (used in the Reynolds number calculation); a typical value is around 1.460E-5 m^2/s for air (wind turbines) and 1.004E-6 m^2/s for seawater (MHK turbines). `SpdSound` is the speed of sound in the fluid (used to calculate the Mach number within the unsteady airfoil aerodynamics calculations); a typical value is around 340.3 m/s for air (wind turbines) and 1500 m/s for seawater (MHK turbines). The last two parameters in this section are only used when `CavitCheck = TRUE` for MHK turbines. `Patm` is the atmospheric pressure above the free surface; typically around 101,325 Pa. `Pvap` is the vapor pressure of the fluid; for seawater this is typically around 2,000 Pa.

4.2.1.3.2.3. Blade-Element/Momentum Theory Options

The input parameters in this section are not used when `WakeMod = 0`.

`SkewMod` determines the skewed-wake correction model. Set `SkewMod` to 1 to use the uncoupled BEM solution technique without an additional skewed-wake correction. Set `SkewMod` to 2 to include the Pitt/Peters correction model. **The coupled model ``SkewMod= 3`` is not available in this version of AeroDyn.**

`SkewModFactor` is used only when `SkewMod = 2`. Enter a scaling factor to use in the Pitt/Peters correction model, or enter `"default"` to use the default value of $\frac{15\pi}{32}$.

Set `TipLoss` to TRUE to include the Prandtl tip-loss model or FALSE to disable it. Likewise, set `HubLoss` to TRUE to include the Prandtl hub-loss model or FALSE to disable it.

Set `TanInd` to TRUE to include tangential induction (from the angular momentum balance) in the BEM solution or FALSE to neglect it. Set `AIDrag` to TRUE to include drag in the axial-induction calculation or FALSE to neglect it. If `TanInd = TRUE`, set `TIDrag` to TRUE to include drag in the tangential-induction calculation or FALSE to neglect it. Even when drag is not used in the BEM iteration, drag is still used to calculate the nodal loads once the induction has been found,

`IndToler` sets the convergence threshold for the iterative nonlinear solve of the BEM solution. The nonlinear solve is in terms of the inflow angle, but `IndToler` represents the tolerance of the nondimensional residual equation, with no physical association possible. When the keyword `DEFAULT` is used in place of a numerical value, `IndToler` will be set to 5E-5 when AeroDyn is compiled in single precision and to 5E-10 when AeroDyn is compiled in double precision; we recommend using these defaults. `MaxIter` determines the maximum number of iterations steps in the BEM solve. If the residual value of the BEM solve is not less than or equal to `IndToler` in `MaxIter`, AeroDyn will exit the BEM solver and return an error message.

4.2.1.3.2.4. Dynamic Blade-Element/Momentum Theory Options

The input parameters in this section are used only when `WakeMod = 2`. The theory is described in Section 4.2.1.7.2.

There are three options available for `DBEMT_Mod`:

- `1`: discrete-time Oye's model, with constant τ_1

- `2` : discrete-time Oye's model, with varying τ_1, automatically adjusted based on inflow. (recommended for time-domain simulations)
- `3` : continuous-time Oye's model, with constant τ_1 (recommended for linearization)

For `DBEMT_Mod=1` or `DBEMT_Mod=3` it is the user responsability to set the value of τ_1 (i.e. `tau1_const`) according to the expression given in Section 4.2.1.7.2, using an estimate of what the mean axial induction $(\overline{a})$ and the mean relative wind velocity across the rotor $(\overline{U_0})$ are for a given simulation.

The option `DBEMT_Mod=3` is the only one that can be used for linearization.

4.2.1.3.2.5. OLAF – cOnvecting LAgrangian Filaments (Free Vortex Wake) Theory Options

The input parameters in this section are used only when `WakeMod = 3`.

The settings for the free vortex wake model are set in the OLAF input file described in Section 4.2.2.4. `OLAFInputFileName` is the filename for this input file.

4.2.1.3.2.6. Unsteady Airfoil Aerodynamics Options

The input parameters in this section are used only when `AFAeroMod = 2`.

`UAMod` determines the UA model. It has the following options:

- `1` : the discrete-time model of Beddoes-Leishman (B-L) (**not currently functional**),
- `2` : the extensions to B-L developed by González (changes in Cn, Cc, Cm)
- `3` : the extensions to B-L developed by Minnema/Pierce (changes in Cc and Cm)
- `4` : 4-states continuous-time B-L model developed by Hansen, Gaunna, and Madsen (HGM). NOTE: might require smaller time steps until a stiff integrator is implemented.
- `5` : 5-states continuous-time B-L model similar to HGM with an additional state for vortex generation
- `6` : 1-state continuous-time developed by Oye
- `7` : discrete-time Boeing-Vertol (BV) model

Linearization is supported with `UAMod=4,5,6` (which use continuous-time states) but not with the other models. The different models are described in Section 4.2.1.8.

While all of the UA models are documented in this manual, the original B-L model is not yet functional. Testing has shown that the González and Minnema/Pierce models produce reasonable hysteresis of the normal force, tangential force, and pitching-moment coefficients if

the UA model parameters are set appropriately for a given airfoil, Reynolds number, and/or Mach number. However, the results will differ a bit from earlier versions of AeroDyn, (which was based on the Minnema/Pierce extensions to B-L) even if the default UA model parameters are used, due to differences in the UA model logic between the versions. We recommend that users run test cases with uniform inflow and fixed yaw error (e.g., through the standalone AeroDyn driver) to examine the accuracy of the normal force, tangential force, and pitching-moment coefficient hysteresis and to adjust the UA model parameters appropriately.

`FLookup` determines how the nondimensional separation distance value, *f'*, will be calculated. When `FLookup` is set to TRUE, *f'* is determined via a lookup into the static lift-force coefficient and drag-force coefficient data. **Using best-fit exponential equations (``FLookup = FALSE``) is not yet available, so ``FLookup`` must be ``TRUE`` in this version of AeroDyn.** Note, `FLookup` is not used when `UAMod=4` or `UAMod=5`.

`UAStartRad` is the starting rotor radius where dynamic stall will be turned on. Enter a number between 0 and 1, representing a fraction of rotor radius, to indicate where unsteady aerodynamics should begin turning on. If this line is omitted from the input file, `UAStartRad` will default to 0 (turning on at the blade root). All blade nodes that are located at a rotor radius less than `UAStartRad` will have unsteady aerodynamics turned off for the entire simulation.

`UAEndRad` is the ending rotor radius where dynamic stall will be turned on. Enter a number between 0 and 1, representing a fraction of rotor radius, to indicate the last rotor radius where unsteady aerodynamics should be turned on. If this line is omitted from the input file, `UAEndRad` will default to 1 (the blade tip). All blade nodes that are located at a rotor radius greater than `UAEndRad` will have unsteady aerodynamics turned off for the entire simulation.

4.2.1.3.2.7. Airfoil Information

This section defines the airfoil data input file information. The airfoil data input files themselves (one for each airfoil) include tables containing coefficients of lift force, drag force, and optionally pitching moment, and minimum pressure versus AoA, as well as UA model parameters, and are described in Section 4.2.1.3.3.

The first 5 lines in the AIRFOIL INFORMATION section relate to the format of the tables of static airfoil coefficients within each of the airfoil input files. `InCol_Alfa`, `InCol_Cl`, `InCol_Cd`, `InCol_Cm,` and `InCol_Cpmin` are column numbers in the tables containing the AoA, lift-force coefficient, drag-force coefficient, pitching-moment coefficient, and minimum pressure coefficient, respectively (normally these are 1, 2, 3, 4, and 5, respectively). If pitching-moment terms are neglected with `UseBlCm = FALSE`, `InCol_Cm` may be set to zero, and if the cavitation check is disabled with `CavitCheck = FALSE`, `InCol_Cpmin` may be set to zero.

Specify the number of airfoil data input files to be used using `NumAFfiles`, followed by `NumAFfiles` lines of filenames. The file names should be in quotations and can contain an absolute path or a relative path e.g., “C:\airfoils\S809_CLN_298.dat” or “airfoils\S809_CLN_298.dat”. If you use relative paths, it is relative to the location of the file in which it is specified. The blade data input files will reference these airfoil data using their line identifier, where the first airfoil file is numbered 1 and the last airfoil file is numbered `NumAFfiles`.

4.2.1.3.2.8. Rotor/Blade Properties

Set `UseBlCm` to TRUE to include pitching-moment terms in the blade airfoil aerodynamics or FALSE to neglect them; if `UseBlCm = TRUE`, pitching-moment coefficient data must be included in the airfoil data tables with `InCol_Cm` not equal to zero.

The blade nodal discretization, geometry, twist, chord, airfoil identifier, and buoyancy properties are set in separate input files for each blade, described in Section 4.2.1.3.4. `ADBlFile(1)` is the filename for blade 1, `ADBlFile(2)` is the filename for blade 2, and `ADBlFile(3)` is the filename for blade 3, respectively; the latter is not used for two-bladed rotors and the latter two are not used for one-bladed rotors. The file names should be in quotations and can contain an absolute path or a relative path. The data in each file need not be identical, which permits modeling of aerodynamic imbalances.

4.2.1.3.2.9. Hub Properties

The input parameters in this section pertain to the calculation of buoyant loads on the hub and are only used when `Buoyancy = TRUE`.

`VolHub` is the volume of the hub and `HubCenBx` is the x offset of the hub center of buoyancy from the hub center in local hub coordinates; offsets in the y and z directions are assumed to be zero. To neglect buoyant loads on the hub, set `VolHub` to 0.

Since the hub and blades are joined elements, hub buoyancy should be turned on if blade buoyancy is on, and vice versa.

4.2.1.3.2.10. Nacelle Properties

The input parameters in this section pertain to the calculation of buoyant loads on the nacelle and are only used when `Buoyancy = TRUE`.

`VolNac` is the volume of the nacelle and ``NacCenB` `` is the position (x,y,z vector) of the nacelle center of buoyancy from the yaw bearing in local nacelle coordinates. To neglect buoyant loads on the nacelle, set `VolNac` to 0.

4.2.1.3.2.11. Tail fin AeroDynamics

The tail fin aerodynamics section contains two lines:

```
======  Tail fin AeroDynamics
========================================================================
    true       TFinAero     - Calculate tail fin aerodynamics model (flag)
  ""           TFinFile     - Input file for tail fin aerodynamics [used only when
TFinAero=True]
======  Tower Influence and Aerodynamics
=============================================================
```

TFinAero Flag to activate the tail fin aerodynamics calculation.

TFinFile Path (absolute or relative to the AeroDyn input file) where the tail fin input file is located.

The content of the tail fin input file is described in Section 4.2.1.3.5.

4.2.1.3.2.12. Tower Influence and Aerodynamics

The input parameters in this section pertain to the tower influence, tower drag, and/or tower buoyancy calculations and are only used when `TwrPotent` > 0, `TwrShadow` > 0, `TwrAero = TRUE`, or `Buoyancy = TRUE`.

`NumTwrNds` is the user-specified number of tower analysis nodes and determines the number of rows in the subsequent table (after two table header lines). `NumTwrNds` must be greater than or equal to two; the higher the number, the finer the resolution and longer the computational time; we recommend that `NumTwrNds` be between 10 and 20 to balance accuracy with computational expense. For each node, `TwrElev` specifies the local elevation of the tower node above ground (or relative to MSL for offshore wind and floating MHK turbines or relative to the seabed for fixed MHK turbines), `TwrDiam` specifies the local tower diameter, `TwrCd` specifies the local tower drag-force coefficient, `TwrTI` specifies the turbulence intensity used in the Eames tower shadow model (`TwrShadow` = 2) as a fraction (rather than a percentage) of the wind fluctuation, and `TwrCb` specifies the tower buoyancy coefficient. `TwrElev` must be entered in monotonically increasing order—from the lowest (tower-base) to the highest (tower-top) elevation. For floating MHK turbines with the tower below MSL, tower nodes should be entered as increasingly negative values, from the tower-base (closest to the platform) to the tower-top (closest to the nacelle). Values of `TwrTI` between 0.05 and 0.4 are recommended. Values larger than 0.4 up to 1 will trigger a warning that the results will need to be interpreted carefully, but the code will allow such values for scientific investigation purposes. `TwrCb` is defined at each node as the cross-sectional area of the tower divided by the area of a circle with diameter equal to the characteristic length of the tower cross section (i.e., `TwrDiam`). For towers with circular cross-sections, `TwrCb` will likely be 1.0 at each node. To neglect buoyant loads on the tower, set `TwrCb` to 0. See Fig. 4.4.

4.2.1.3.2.13. Outputs

Specifying `SumPrint` to TRUE causes AeroDyn to generate a summary file with name `<OutFileRoot>.AD.sum`. `<OutFileRoot>` is either specified in the I/O SETTINGS section of the driver input file when running AeroDyn standalone, or by the OpenFAST program when running a coupled simulation. See Section 4.2.1.4.2 for summary file details. If `AFAeroMod=2`, the unsteady aero module will also generate a file called `<OutFileRoot>.UA.sum` that will list all of the UA parameters used in the airfoil tables. This allows the user to check what values are being used in case the code has computed the parameters without user input.

AeroDyn can output aerodynamic and kinematic quantities at up to nine nodes specified along the tower and up to nine nodes along each blade. For outputs at every blade node, see Section 4.2.1.3.2.14.

`NBlOuts` specifies the number of blade nodes that output is requested for (0 to 9) and `BlOutNd` on the next line is a list `NBlOuts` long of node numbers between 1 and `NumBlNds` (corresponding to a row number in the blade analysis node table in the blade data input files), separated by any combination of commas, semicolons, spaces, and/or tabs. All blades have the same output node numbers. `NTwOuts` specifies the number of tower nodes that output is requested for (0 to 9) and `TwOutNd` on the next line is a list `NTwOuts` long of node numbers between 1 and `NumTwrNds` (corresponding to a row number in the tower analysis node table above), separated by any combination of commas, semicolons, spaces, and/or tabs. The outputs specified in the `OutList` section determine which quantities are actually output at these nodes.

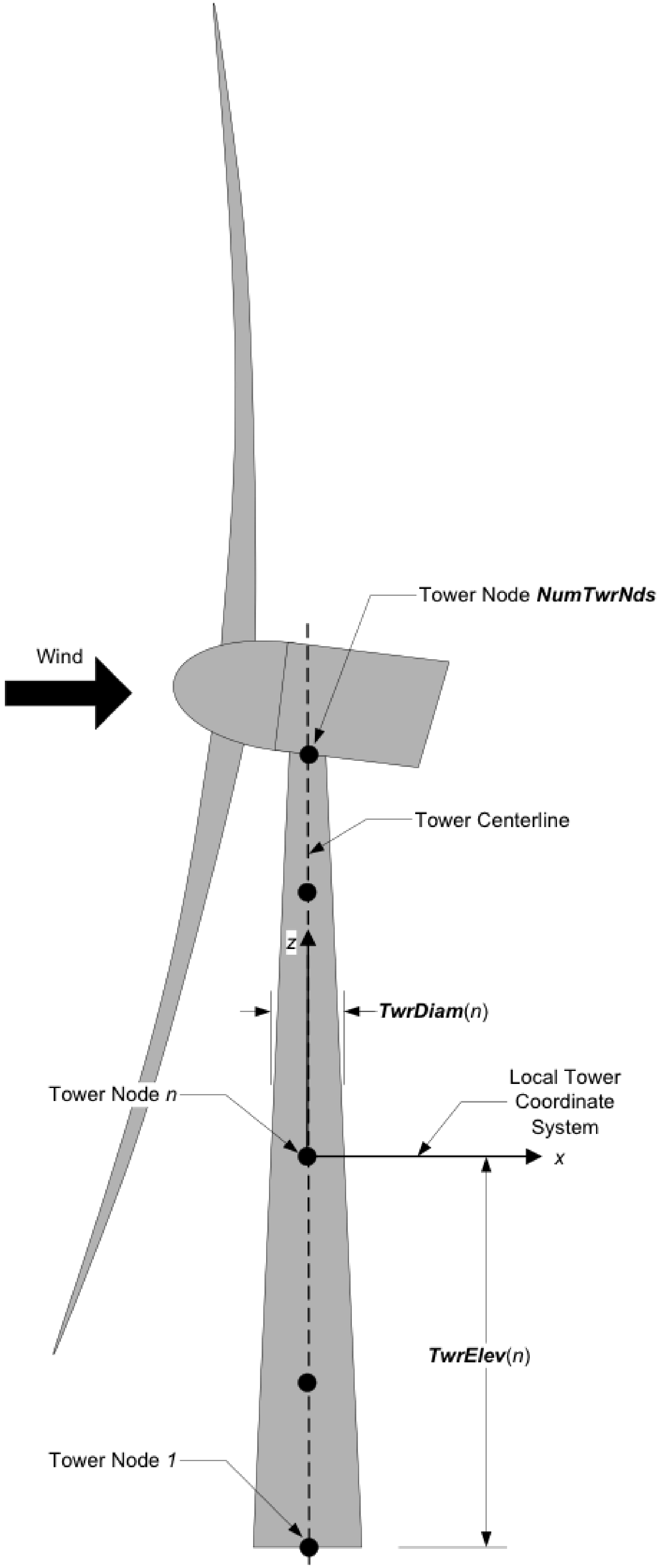

Fig. 4.4 *AeroDyn Tower Geometry*

The `OutList` section controls output quantities generated by AeroDyn. Enter one or more lines containing quoted strings that in turn contain one or more output parameter names. Separate output parameter names by any combination of commas, semicolons, spaces, and/or tabs. If you prefix a parameter name with a minus sign, "-", underscore, "_", or the characters "m" or "M", AeroDyn will multiply the value for that channel by –1 before writing the data. The parameters are written in the order they are listed in the input file. AeroDyn allows you to use multiple lines so that you can break your list into meaningful groups and so the lines can be shorter. You may enter comments after the closing quote on any of the lines. Entering a line with the string "END" at the beginning of the line or at the beginning of a quoted string found at the beginning of the line will cause AeroDyn to quit scanning for more lines of channel names. Blade and tower node-related quantities are generated for the requested nodes identified through the `BlOutNd` and `TwOutNd` lists above. If AeroDyn encounters an unknown/invalid channel name, it warns the users but will remove the suspect channel from the output file. Please refer to Appendix E for a complete list of possible output parameters.

4.2.1.3.2.14. Nodal Outputs

In addition to the named outputs in Section 4.2.1.3.2.13 above, AeroDyn allows for outputting the full set blade node motions and loads (tower nodes unavailable at present). Please refer to the AeroDyn_Nodes tab in the Excel file OutListParameters.xlsx for a complete list of possible output parameters.

This section follows the *END* statement from normal Outputs section described above, and includes a separator description line followed by the following options.

BldNd_BladesOut specifies the number of blades to output. Possible values are 0 through the number of blades AeroDyn is modeling. If the value is set to 1, only blade 1 will be output, and if the value is 2, blades 1 and 2 will be output.

BldNd_BlOutNd specifies which nodes to output. This is currently unused.

The **OutList** section controls the nodal output quantities generated by AeroDyn. In this section, the user specifies the name of the channel family to output. The output name for each channel is then created internally by AeroDyn by combining the blade number, node number, and channel family name. For example, if the user specifies **AxInd** as the channel family name, the output channels will be named with the convention of **BβN###AxInd** where β is the blade number, and **###** is the three digit node number.

4.2.1.3.2.14.1. Sample Nodal Outputs section

This sample includes the `END` statement from the regular outputs section.

```
END of input file (the word "END" must appear in the first 3 columns of this last OutList line)
---------------------- NODE OUTPUTS ------------------------------------------
          3   BldNd_BladesOut  - Blades to output
         99   BldNd_BlOutNd   - Blade nodes on each blade (currently unused)
              OutList     - The next line(s) contains a list of output parameters. See OutListParameters.xlsx, AeroDyn_Nodes tab for a listing of available output channels, (-)
"VUndx"     - x-component of undisturbed wind velocity at each node
"VUndy"     - y-component of undisturbed wind velocity at each node
"VUndz"     - z-component of undisturbed wind velocity at each node
"VDisx"     - x-component of disturbed wind velocity at each node
"VDisy"     - y-component of disturbed wind velocity at each node
"VDisz"     - z-component of disturbed wind velocity at each node
"STVx"      - x-component of structural translational velocity at each node
"STVy"      - y-component of structural translational velocity at each node
"STVz"      - z-component of structural translational velocity at each node
"VRel"      - Relvative wind speed at each node
"DynP"      - Dynamic pressure at each node
"Re"        - Reynolds number (in millions) at each node
"M"         - Mach number at each node
"Vindx"     - Axial induced wind velocity at each node
"Vindy"     - Tangential induced wind velocity at each node
"AxInd"     - Axial induction factor at each node
"TnInd"     - Tangential induction factor at each node
"Alpha"     - Angle of attack at each node
"Theta"     - Pitch+Twist angle at each node
"Phi"       - Inflow angle at each node
"Curve"     - Curvature angle at each node
"Cl"        - Lift force coefficient at each node
"Cd"        - Drag force coefficient at each node
"Cm"        - Pitching moment coefficient at each node
"Cx"        - Normal force (to plane) coefficient at each node
"Cy"        - Tangential force (to plane) coefficient at each node
"Cn"        - Normal force (to chord) coefficient at each node
"Ct"        - Tangential force (to chord) coefficient at each node
"Fl"        - Lift force per unit length at each node
"Fd"        - Drag force per unit length at each node
"Mm"        - Pitching moment per unit length at each node
"Fx"        - Normal force (to plane) per unit length at each node
"Fy"        - Tangential force (to plane) per unit length at each node
"Fn"        - Normal force (to chord) per unit length at each node
"Ft"        - Tangential force (to chord) per unit length at each node
"Clrnc"     - Tower clearance at each node (based on the absolute distance to the nearest point in the tower from blade node B#N# minus the local tower radius, in the deflected configuration); please note that this clearance is only approximate because the calculation assumes that the blade is a line with no volume (however, the calculation does use the local tower radius); when blade node B#N# is above the tower top (or below the tower base), the absolute distance to the tower top (or base) minus the local tower radius, in the deflected configuration, is output
"Vx"        - Local axial velocity
"Vy"        - Local tangential velocity
"GeomPhi"   - Geometric phi? If phi was solved using normal BEMT equations, GeomPhi = 1; otherwise, if it was solved geometrically, GeomPhi = 0.
"Chi"       - Skew angle (used in skewed wake correction) -- not available for OLAF
"UA_Flag"   - Flag indicating if UA is turned on for this node. -- not available for OLAF
"CpMin"     - Pressure coefficient
"SgCav"     - Cavitation number
```

```
"SigCr"     - Critical cavitation number
"Gam"       - Gamma -- circulation on blade
"Cl_Static" - Static portion of lift force coefficient at each node, without unsteady effects -- not available for BEMT/DBEMT
"Cd_Static" - Static portion of drag force coefficient at each node, without unsteady effects -- not available for BEMT/DBEMT
"Cm_Static" - Static portion of pitching moment coefficient at each node, without unsteady effects -- not available for BEMT/DBEMT
"Uin"       - Axial induced velocity in rotating hub coordinates. Axial aligned with hub axis.     rotor plane polar hub rotating coordinates
"Uit"       - Tangential induced velocity in rotating hub coordinates. Tangential to the rotation plane. Perpendicular to blade aziumth.     rotor plane polar hub rotating coordinates
"Uir"       - Radial induced velocity in rotating hub coordinates. Radial outwards in rotation plane. Aligned with blade azimuth.     rotor plane polar hub rotating coordinates
"Fbn"       - Buoyant force normal to chord per unit length at each node
"Fbt"       - Buoyant force tangential to chord per unit length at each node
"Fbs"       - Buoyant spanwise force per unit length at each node
"Mbn"       - Buoyant moment normal to chord per unit length at each node
"Mbt"       - Buoyant moment tangential to chord per unit length at each node
"Mbs"       - Buoyant spanwise moment per unit length at each node
END of input file (the word "END" must appear in the first 3 columns of this last OutList line)
---------------------------------------------------------------------------------------
```

4.2.1.3.2.15. Tail fin outputs

The tail fin outputs are:

- TFinAlpha (deg): Angle of attack at the reference point of the fin
- TFinDynP (Pa): Dynamic pressure at the reference point of the fin
- TFinM (-): Mach number at the reference point of the fin
- TFinRe (-): Reynolds number at the reference point of the fin
- TFinVrel (m/s): Orthogonal relative velocity norm ($V_{rel,\perp}$) at the reference point of the fin
- TFinVdisxi (m/s): Disturbed velocity (x) at the reference point of the fin in the inertial coordinate system
- TFinVdisyi (m/s): Disturbed velocity (y) at the reference point of the fin in the inertial coordinate system
- TFinVdiszi (m/s): Disturbed velocity (z) at the reference point of the fin in the inertial coordinate system
- TFinVrelxi (m/s): Relative velocity (x) at the reference point of the fin in the inertial coordinate system
- TFinVrelyi (m/s): Relative velocity (y) at the reference point of the fin in the inertial coordinate system
- TFinVrelzi (m/s): Relative velocity (z) at the reference point of the fin in the inertial coordinate system
- TFinVundxi (m/s): Undisturbed velocity (x) at the reference point of the fin in the inertial coordinate system
- TFinVundyi (m/s): Undisturbed velocity (y) at the reference point of the fin in the inertial coordinate system
- TFinVundzi (m/s): Undisturbed velocity (z) at the reference point of the fin in the inertial coordinate system
- TFinSTVxi (m/s): Structural velocity (x) at the reference point of the fin in the inertial coordinate system
- TFinSTVyi (m/s): Structural velocity (y) at the reference point of the fin in the inertial coordinate system
- TFinSTVzi (m/s): Structural velocity (z) at the reference point of the fin in the inertial coordinate system
- TFinFxi (N) : Aerodynamic force (x) at the reference point of the fin in the inertial coordinate system
- TFinFyi (N) : Aerodynamic force (y) at the reference point of the fin in the inertial coordinate system
- TFinFzi (N) : Aerodynamic force (z) at the reference point of the fin in the inertial coordinate system
- TFinMxi (Nm): Aerodynamic moment (x) at the reference point of the fin in the inertial coordinate system
- TFinMyi (Nm): Aerodynamic moment (y) at the reference point of the fin in the inertial coordinate system
- TFinMzi (Nm): Aerodynamic moment (z) at the reference point of the fin in the inertial coordinate system

4.2.1.3.3. Airfoil Data Input File

The airfoil data input files themselves (one for each airfoil) include tables containing coefficients of lift force, drag force, and pitching moment versus AoA, as well as UA model parameters. In these files, any line whose first non-blank character is an exclamation point (!) is ignored (for inserting comment lines). The non-comment lines should appear within the file in order, but comment lines may be intermixed as desired for reading clarity. A sample airfoil data input file is given in Section 4.2.1.11.

`InterpOrd` is the order the static airfoil data is interpolated when AeroDyn uses table look-up to find the lift-, drag-, and optional pitching-moment, and minimum pressure coefficients as a function of AoA. When `InterpOrd` is 1, linear interpolation is used; when `InterpOrd` is 3, the data will be interpolated with cubic splines; if the keyword `DEFAULT` is entered in place of a numerical value, `InterpOrd` is set to 1.

`RelThickness` is the non-dimensional thickness of the airfoil (thickness over chord ratio), expressed as a fraction (not a percentage), typically between 0.1 and 1. The parameter is currently used when `UAMod=7`, but might be used more in the future. The default value of 0.2 if provided for convenience.

`NonDimArea` is the nondimensional airfoil area (normalized by the local `BlChord` squared), but is currently unused by AeroDyn.

`NumCoords` is the number of points to define the exterior shape of the airfoil, plus one point to define the aerodynamic center, and determines the number of rows in the subsequent table; `NumCoords` must be exactly zero or greater than or equal to three. For each point, the nondimensional *X* and *Y* coordinates are specified in the table, `X_Coord` and `Y_Coord` (normalized by the local `BlChord`). The first point must always locate the aerodynamic center (reference point for the airfoil lift and drag forces, likely not on the surface of the airfoil); the remaining points should define the exterior shape of the airfoil. The airfoil shape is currently unused by AeroDyn, but when AeroDyn is coupled to OpenFAST, the airfoil shape will be used by OpenFAST for blade surface visualization when enabled.

`BL_file` is the name of the file containing boundary-layer characteristics of the profile. It is ignored if the aeroacoustic module is not used. This parameter may also be omitted from the file if the aeroacoustic module is not used.

Specify the number of Reynolds number- or aerodynamic-control setting-dependent tables of data for the given airfoil via the `NumTabs` setting. The remaining parameters in the airfoil data input files are entered separately for each table.

`Re` and `UserProp` are the Reynolds number (in millions) and aerodynamic-control (or user property) setting for the included table. These values are used only when the `AFTabMod` parameter in the primary AeroDyn input file is set to use 2D interpolation based on `Re` or `UserProp`. If 1D interpolation (based only on angle of attack) is used, only the first table in the file will be used.

Set `InclUAdata` to TRUE if you are including the UA model parameters. If this is set to FALSE, all of the UA model parameters will be determined by the code. Any lines that are missing from this section will have their values determined by the code, either using a default value or calculating it based on the polar coefficient data in the airfoil table:

- `alpha0` specifies the zero-lift AoA (in degrees);
- `alpha1` specifies the AoA (in degrees) larger than `alpha0` for which *f* equals 0.7; approximately the positive stall angle; This parameter is used when `flookup=false` and when determining a default value of `Cn1`.
- `alpha2` specifies the AoA (in degrees) less than `alpha0` for which *f* equals 0.7; approximately the negative stall angle; This parameter is used when `flookup=false` and when determining a default value of `Cn2`.
- `alphaUpper` specifies the AoA (in degrees) of the upper boundary of fully-attached region of the cn or cl curve. It is used to compute the separation function when `UAMod=5`.
- `alphaLower` specifies the AoA (in degrees) of the lower boundary of fully-attached region of the cn or cl curve. It is used to compute the separation function when `UAMod=5`. (The separation function will have a value of 1 between `alphaUpper` and `alphaLower`.)
- `eta_e` is the recovery factor and typically has a value in the range [0.85 to 0.95] for `UAMod = 1`; if the keyword `DEFAULT` is entered in place of a numerical value, `eta_e` is set to 0.9 for `UAMod = 1`, but `eta_e` is set to 1.0 for other `UAMod` values and whenever `FLookup = TRUE`;
- `C_nalpha` is the slope of the 2D normal force coefficient curve in the linear region;
- `C_lalpha` is the slope of the 2D normal lift coefficient curve in the linear region; Used for `UAMod=4,6`.
- `T_f0` is the initial value of the time constant associated with *Df* in the expressions of *Df* and *f'*; if the keyword `DEFAULT` is entered in place of a numerical value, `T_f0` is set to 3.0;
- `T_V0` is the initial value of the time constant associated with the vortex lift decay process, used in the expression of `Cvn`; it depends on Reynolds number, Mach number, and airfoil; if the keyword `DEFAULT` is entered in place of a numerical value, `T_V0` is set to 6.0;
- `T_p` is the boundary-layer leading edge pressure gradient time constant in the expression for *Dp* and should be tuned based on airfoil experimental data; if the keyword `DEFAULT` is entered in place of a numerical value, `T_p` is set to 1.7;

- `T_VL` is the time constant associated with the vortex advection process, representing the nondimensional time in semi-chords needed for a vortex to travel from the leading to trailing edges, and used in the expression of *Cvn*; it depends on Reynolds number, Mach number (weakly), and airfoil; valued values are in the range [6 to 13]; if the keyword `DEFAULT` is entered in place of a numerical value, `T_VL` is set to 11.0;
- `b1` is a constant in the expression of φ_α^c and φ_q^c; this value is relatively insensitive for thin airfoils, but may be different for turbine airfoils; if the keyword `DEFAULT` is entered in place of a numerical value, `b1` is set to 0.14, based on experimental results;
- `b2` is a constant in the expression of φ_α^c and φ_q^c; this value is relatively insensitive for thin airfoils, but may be different for turbine airfoils; if the keyword `DEFAULT` is entered in place of a numerical value, `b2` is set to 0.53, based on experimental results;
- `b5` is a constant in the expression of K_q''', Cm_q^{nc}, and K_{m_q}; if the keyword `DEFAULT` is entered in place of a numerical value, `b5` is set to 5, based on experimental results;
- `A1` is a constant in the expression φ_α^c and φ_q^c; this value is relatively insensitive for thin airfoils, but may be different for turbine airfoils; if the keyword `DEFAULT` is entered in place of a numerical value, `A1` is set to 0.3, based on experimental results;
- `A2` is a constant in the expression φ_α^c and φ_q^c; this value is relatively insensitive for thin airfoils, but may be different for turbine airfoils; if the keyword `DEFAULT` is entered in place of a numerical value, `A2` is set to 0.7, based on experimental results;
- `A5` is a constant in the expression K_q''', Cm_q^{nc}, and K_{m_q}; if the keyword `DEFAULT` is entered in place of a numerical value, `A5` is set to 1, based on experimental results;
- `S1` is the constant in the best fit curve of *f* for `alpha0` ≤ AoA ≤ `alpha1` for `UAMod = 1` (and is unused otherwise); by definition, it depends on the airfoil;
- `S2` is the constant in the best fit curve of *f* for AoA > `alpha1` for `UAMod = 1` (and is unused otherwise); by definition, it depends on the airfoil;
- `S3` is the constant in the best fit curve of *f* for `alpha2` ≤ AoA ≤ `alpha0` for `UAMod = 1` (and is unused otherwise); by definition, it depends on the airfoil;
- `S4` is the constant in the best fit curve of *f* for AoA < `alpha2` for `UAMod = 1` (and is unused otherwise); by definition, it depends on the airfoil;
- `Cn1` is the critical value of C_n' at leading-edge separation for positive AoA and should be extracted from airfoil data at a given Reynolds number and Mach number; `Cn1` can be calculated from the static value of *Cn* at either the break in the pitching moment or the loss of chord force at the onset of stall; `Cn1` is close to the condition of maximum lift of the airfoil at low Mach numbers;
- `Cn2` is the critical value of C_n' at leading-edge separation for negative AoA and should be extracted from airfoil data at a given Reynolds number and Mach number; `Cn2` can be calculated from the static value of *Cn* at either the break in the pitching moment or the loss of chord force at the onset of stall; `Cn2` is close to the condition of maximum lift of the airfoil at low Mach numbers;

- `St_sh` is the Strouhal's shedding frequency; if the keyword `DEFAULT` is entered in place of a numerical value, `St_sh` is set to 0.19;
- `Cd0` is the drag-force coefficient at zero-lift AoA;
- `Cm0` is the pitching-moment coefficient about the quarter-chord location at zero-lift AoA, positive for nose up;
- `k0` is a constant in the best fit curve of $\hat{x_{cp}}$ and equals for $\hat{x_{AC}} - 0.25$ `UAMod = 1` (and is unused otherwise);
- `k1` is a constant in the best fit curve of $\hat{x_{cp}}$ for `UAMod = 1` (and is unused otherwise);
- `k2` is a constant in the best fit curve of $\hat{x_{cp}}$ for `UAMod = 1` (and is unused otherwise);
- `k3` is a constant in the best fit curve of $\hat{x_{cp}}$ for `UAMod = 1` (and is unused otherwise);
- `k1_hat` is a constant in the expression of *Cc* due to leading-edge vortex effects for `UAMod = 1` (and is unused otherwise);
- `x_cp_bar` is a constant in the expression of $\hat{x_{cp}^{\nu}}$ for `UAMod = 1` (and is unused otherwise); if the keyword `DEFAULT` is entered in place of a numerical value, `x_cp_bar` is set to 0.2; and
- `UACutOut` is the AoA (in degrees) in absolute value above which UA are disabled; if the keyword `DEFAULT` is entered in place of a numerical value, `UACutOut` is set to 45.
- `UACutOut_delta` is the AoA difference (in degrees) which, together with `UACutOut` determines when unsteady aero begins to turn off; if the keyword `DEFAULT` is entered in place of a numerical value, `UACutOut_delta` is set to 5 degrees. The unsteady solution is used at angles of attack less than `UACutOut - UACutout_delta` degrees. Above `UACutout`, the steady solution is used (i.e., UA is disabled). The steady and unsteady solutions are blended between those two angles.
- `filtCutOff` is the cut-off reduced frequency of the low-pass filter applied to the AoA input to UA, as well as to the pitch rate and pitch acceleration derived from AoA within UA; if the keyword `DEFAULT` is entered in place of a numerical value, `filtCutOff` is set to 0.5. This non-dimensional value corresponds to a frequency of $\frac{U \times \mathrm{filtCutOff}}{\pi \times \mathrm{chord}}$ Hz.

`NumAlf` is the number of distinct AoA entries and determines the number of rows in the subsequent table of static airfoil coefficients; `NumAlf` must be greater than or equal to one (`NumAlf = 1` implies constant coefficients, regardless of the AoA).

AeroDyn will interpolate on AoA using the data provided via linear interpolation or via cubic splines, depending on the setting of input `InterpOrd` above. If `AFTabMod` is set to `1`, only the first airfoil table in each file will be used. If `AFTabMod` is set to `2`, AeroDyn will find the airfoil tables that bound the computed Reynolds number, and linearly interpolate between the tables, using the logarithm of the Reynolds numbers. If `AFTabMod` is set to `3`, it will find the bounding airfoil tables based on the `UserProp` field and linearly interpolate the tables based on it. Note that OpenFAST currently sets the `UserProp` input value to `0` unless the DLL controller is used and sets the value, so using this feature may require a code change.

For each AoA, you must set the AoA (in degrees), `alpha`, the lift-force coefficient, `Coefs`(:,1), the drag-force coefficient, `Coefs(:,2)`, and optionally the pitching-moment coefficient, `Coefs(:,3)`, and minimum pressure coefficient, `Coefs(:,4)`, but the column order depends on the settings of `InCol_Alfa`, `InCol_Cl`, `InCol_Cd`, `InCol_Cm`, and `InCol_Cpmin` in the AIRFOIL INFORMATION section of the AeroDyn primary input file. AoA must be entered in monotonically increasing order—from lowest to highest AoA; the first row should be for AoA = –180 degrees and the last should be for AoA = +180 (unless `NumAlf = 1`, in which case AoA is unused). If pitching-moment terms are neglected with `UseBlCm = FALSE` in the ROTOR/BLADE PROPERTIES section of the AeroDyn primary input file, the column containing pitching-moment coefficients may be absent from the file. Likewise, if the cavitation check is neglected with `CavitCheck = FALSE` in the GENERAL OPTIONS section of the AeroDyn primary input file, the column containing the minimum pressure coefficients may be absent from the file.

4.2.1.3.4. Blade Data Input File

The blade data input file contains the nodal discretization, geometry, twist, chord, airfoil identifier, and buoyancy properties for a blade. Separate files are used for each blade, which permits modeling of aerodynamic imbalances. A sample blade data input file is given in Section 4.2.1.11.

The input file begins with two lines of header information which is for your use, but is not used by the software.

`NumBlNds` is the user-specified number of blade analysis nodes and determines the number of rows in the subsequent table (after two table header lines). `NumBlNds` must be greater than or equal to two; the higher the number, the finer the resolution and longer the computational time; we recommend that `NumBlNds` be between 10 and 20 to balance accuracy with computational expense. Even though `NumBlNds` is defined in each blade file, all blades must have the same number of nodes. For each node:

- `BlSpn` specifies the local span of the blade node along the (possibly preconed) blade-pitch axis from the root; `BlSpn` must be entered in monotonically increasing order—from the most inboard to the most outboard—and the first node must be zero, and when AeroDyn is coupled to OpenFAST, the last node should be located at the blade tip;
- `BlCrvAC` specifies the local out-of-plane offset (when the blade-pitch angle is zero) of the aerodynamic center (reference point for the airfoil lift and drag forces), normal to the blade-pitch axis, as a result of blade curvature; `BlCrvAC` is positive downwind; upwind turbines have negative `BlCrvAC` for improved tower clearance;
- `BlSwpAC` specifies the local in-plane offset (when the blade-pitch angle is zero) of the aerodynamic center (reference point for the airfoil lift and drag forces), normal to the blade-pitch axis, as a result of blade sweep; positive `BlSwpAC` is opposite the direction of rotation;

- `BlCrvAng` specifies the local angle (in degrees) from the blade-pitch axis of a vector normal to the plane of the airfoil, as a result of blade out-of-plane curvature (when the blade-pitch angle is zero); `BlCrvAng` is positive downwind; upwind turbines have negative `BlCrvAng` for improved tower clearance;
- `BlTwist` specifies the local aerodynamic twist angle (in degrees) of the airfoil; it is the orientation of the local chord about the vector normal to the plane of the airfoil, positive to feather, leading edge upwind; the blade-pitch angle will be added to the local twist;
- `BlChord` specifies the local chord length;
- `BlAFID` specifies which airfoil data the local blade node is associated with; valid values are numbers between 1 and `NumAFfiles` (corresponding to a row number in the airfoil file table in the AeroDyn primary input file); multiple blade nodes can use the same airfoil data;
- `BlCb` specifies the blade buoyancy coefficient, defined as the local cross-sectional area of the blade divided by the area of a circle with diameter equal to `BlChord`; to neglect buoyant loads on the blade, set `BlCb` to 0; since the blades and hub are joined elements, blade buoyancy should be turned on if hub buoyancy is on, and vice versa;
- `BlCenBn` specifies the offset of the blade center of buoyancy from the aerodynamic center in the direction normal to the chord (positive pointing toward the suction side of the blade); and
- `BlCenBt` specifies the offset of the blade center of buoyancy from the aerodynamic center in the direction tangential to the chord (positive pointing toward the trailing edge of the blade).

See Fig. 4.5. Twist is shown in Fig. 4.3 of Section 4.2.1.11.

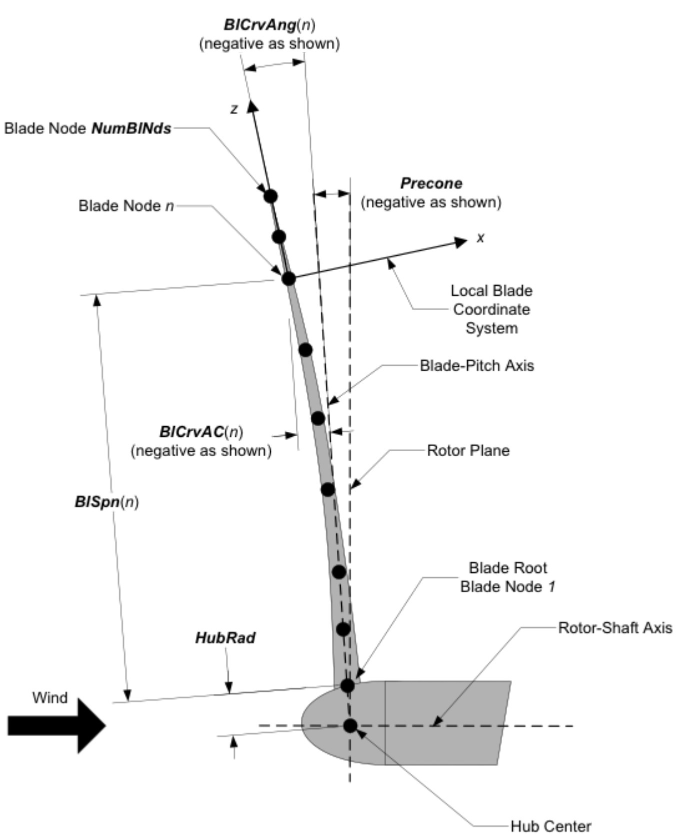

Fig. 4.5 *AeroDyn Blade Geometry – Left: Side View; Right: Front View (Looking Downwind)*

4.2.1.3.5. Tail fin input file

An example of tail fin input file is given below:

```
------- TAIL FIN AERODYNAMICS INPUT FILE-------------------------------------------
Comment
======  General inputs ============================================================
1         TFinMod     - Tail fin aerodynamics model {0: none, 1: polar-based, 2: USB-
based} (switch)
0.5       TFinChord   - Tail fin chord (m) [used only when TFinMod=1]
0.3       TFinArea    - Tail fin planform area (m^2) [used only when TFinMod=1]
10.,0.,0. TFinRefP_n  - Undeflected position of the tail fin reference point wrt the tower
top (m)
0.,0.,0.  TFinAngles  - Tail fin chordline skew, tilt, and bank angles about the
reference point (degrees)
0         TFinIndMod  - Model for induced velocity calculation {0: none, 1:rotor-average}
(switch)
====== Polar-based model ================================ [used only when TFinMod=1]
1        TFinAFID - Index of Tail fin airfoil number [1 to NumAFfiles]
====== Unsteady slender body model  ===================== [used only when TFinMod=2]
0.9           TFinKp        - Tail fin moment of area about reference point
0.3,0.1,0.1   TFinSigma     - Tail fin empirical constant for vortex separation functions
40,60,60      TFinAStar     - Tail fin initial angles for vortex separation functions
(deg)
3.1416        TFinKv        - Tail fin vortex lift coefficient
1.3           TFinCDc       - Tail fin drag coefficient
```

4.2.1.3.5.1. General inputs

`TFinMod` is a switch to select a model for the tail fin aerodynamics: 0) none (the aerodynamic forces are zero), 1) polar-based, 2) USB-based (see Section 4.2.1.9). (switch)

`TFinArea` is the area of the tail fin. (m^2) This is the plan form area of the tail fin plate used to relate the local dynamic pressure and airfoil coefficients to aerodynamic loads. This value must not be negative and is only used when TFinMod is set to 1. (m^2)

`TFinRefP_n` is the undeflected position (x_{ref,x_n}, x_{ref,y_n}, x_{ref,z_n}) of the tail fin from the tower top in nacelle coordinates. (formerly defined using `TFinCPxn`, `TFinCPyn`, `TFinCPzn`). The distances defines the configuration for a furl angle of zero. For a typical upwind wind turbine, x_n, is positive downwind, y_n, is positive to the left when looking downwind, and z_n, is positive upward when looking downwind. See Fig. 4.47 and Fig. 4.9. (m)

`TFinAngles` are the angles (θ_{skew}, θ_{tilt}, θ_{bank}) of the tail fin (formerly defined as `TFinSkew`, `TFinTilt`, `TFinBank`). See Fig. 4.47 and Fig. 4.9. These angles define the chordline at a furl angle of zero, where the chordline is assumed to be passing through the reference point. θ_{skew} is the skew angle of the tail fin chordline in the nominally horizontal plane. Positive skew orients the nominal horizontal projection of the tail fin chordline about the z_n-axis. The aforementioned chordline is the chordline passing through the tail fin reference point. This value must be greater than -180 and less than or equal to 180 degrees. θ_{tilt} is the tilt angle of the tail fin chordline from the nominally horizontal plane. This value must be between -90 and 90 degrees (inclusive). Positive tilt means that the trailing edge of the tail fin is higher than the leading edge. θ_{bank} is the bank angle of the tail fin plane about the tail fin chordline. This value must be greater than -180 and less than or equal to 180 degrees. (deg)

`TFinIndMod` Switch to select a model for the calculation of the velocity induced by the rotor and its wake on the tailfin (not the induced velocity from the tailfin wing). The options available are: 0) none (the induced velocity is zero) 1) rotor-average (using the average induced velocity across all blades and blade nodes) (see Section 4.2.1.9). (switch)

4.2.1.3.5.2. Polar-based model inputs

`TFinAFID` This integer tells AeroDyn which of the input airfoil files (`AFNames`) is assigned to the tail fin. For instance, a value of 2 means that the tail fin will use `AFNames(2)` for the local tail fin airfoil. This value must be between 1 and `NumAFfiles` and is only used when TFinMod is set to 1. (-)

4.2.1.3.5.3. Unsteady slender body (USB) model inputs

Refer to Section 4.2.1.9 and [ad-HWS23] for guidance on how to select parameters for the unsteady slender body theory based model.

`TFinKp` Potential lift coefficient for unsteady aerodynamics. `TFinKp` is used to calculate the potential flow contribution to the unsteady aerodynamic force on the tail fin.

`TFinSigma` Tail fin empirical constants characterizing the decay of separation functions used in the unsteady aerodynamic model. The separation functions and their dependence on `TFinSigma` are described in Section 4.2.1.9.

`TFinAStar` Tail fin characteristics angles for separation functions used in the unsteady aerodynamic model. The separation functions and their dependence on `TFinAStar` are described in Section 4.2.1.9.

`TFinKv` Vortex lift coefficient for unsteady aerodynamics. `TFinKv` is used to calculate the vortex flow contribution to the unsteady aerodynamic force on the tail fin.

`TFinCDc` Tail fin drag coefficient used for unsteady aerodynamic model. The drag on the tail fin significantly contributes to the normal force at high yaw angles.