FS2000

FS-Wind Aerodynamic Loading

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2019

Email:support@aes-uk.com

Table of Contents

1.0 Introduction

FSWind is a pre-processor for the generation of aerodynamic loading on framed structures created in FS2000. Loading due to wind action, coatings (gravity) can be included.

The main capabilities of the program are:

- Wind velocity can be defined as a function of depth and specified in a different direction to the wind direction.
- Load coefficients may be specified by individual definition, global definition, definition as a function of Reynolds Number, or definition as a function of elevation.
- Drag coefficients may be defined relative to "Y" & "Z" directions of the element local co-ordinate system.
- Coatings thickness e.g. ice may be individually specified for each element or defined as a function of height.
- Contents density may be specified by individual definition for each element or defined globally. A contents limit in terms of depth can also be specified i.e. depth above which the element will be assumed empty.
- Loading on non-circular hollow sections and solid sections is possible.
- Loading due to no-structural item can be applied as attached nodal areas

The program will read the default model data from the wind and other related data is then defined using the various input forms. There a few requirements when creating model for wind load generation. These are addressed in Section 4.

-o-

2.0 Basic Program Operation - Overview

When started the FSWind program reads the model data files of the default FS2000 model.

Wind and other related data is then defined using the various input forms. Data associated with the wind load evaluation is saved in two file types;

Aerodynamic Model Data Files   Data associated with the physical model

Environmental Load Data Files   Data associated with the environment.

The program can be activated to evaluate the loading. The shear forces, vertical forces and the centroid of force are evaluated .

Loads Cases will be created during evaluation if the Create Load case option is selected.

-o-

2.1 Aerodynamic Model Data

The Aerodynamic Model Data File (file extension ".WIN") is a file that contains data relating to the aerodynamic properties of the model i.e. drag coefficients, coating profiles etc. If a file exists it will be loaded automatically. This file will only exist if saved during earlier sessions in FSWind with the same model.

The data file is an interpreted file that uses the command line instructions given in Section 10. When the file is save within FSWind extra description lines are added to make the file more readable.

It is always recommended when saving to use file extension **.WIN** for this file.

If additional Aerodynamic Model Data File are required e.g. using different profiles on the same model, then the file extension .**WIN*** should be used. This will ensure that all definition data is archived.

-o-

2.2 Environmental Load Data

Data relating to the environment is saved in the Environmental Load Data Files. Generally there will be one for each load case being generated.

These files can be saved using any legal filename and extension. However, to ensure that the file is archived with the model, the name must be the model name and the first character of the extension must be a **!**. The file extension may be used to identify direction i.e. **".!NW"** or **".!SW"**. Always include a file extension when saving.

It is recommended that before Wind Load Cases are created both the Aerodynamic Model Data and the Environmental Load Data be saved.

-o-

3.0 Background Theory

3.1 Structural Loading (Beam elements)

Transverse Loading (Local Y and Z directions)

There are two basic approaches used to evaluate transverse wind loading on inclined structural members. Although the approaches appear quite different the resulting loads within a structure will be similar as will any overturing moments. The methods do differ in that the shear forces from the Projected Method will only be in the direction of the wind whereas in the Component Method transverse shear will be present. In most structures these transverse loads tend to cancel each out and therefore are not that apparent.The non-tube option of the Component Method will always produce the more onerous loading. The more recently introduced Projected Method which uses the Wind Pressure is more consistent with wind loading codes and is now the default method.

Component Method - In this approach the resultant wind is resolved into local element components and these are used to evaluate the forces in the local directions using $\rho u^2 Cd.D.L / 2$.

Wind Interaction Method option parameter = 0. All element are evaluated using the pipe method shown below

The element force evaluation is based on the following approach. The wind velocity vector is resolved into components parallel and tangential to the element local axis (local x axis).

$$U = (vn^2 + vt^2)^{1/2}$$

The normal component is then resolved into components in the local y and z axis.

$$vn = (vy^2 + vz^2)^{1/2}$$

The local normal force on the element is evaluated by the following.

(only y direction shown)

For pipes $Fy = CDy.\rho.Dz.vnuy/2$

For non pipes $Fy = CDy.\rho.Dz.U.uy/2$

Where:

CDy = Drag coefficient for local y component of flow normal to the member

= Fluid mass density

Dz = Effective member width in local z direction

U $= (vn^2 + vt^2)^{1/2}$ = Resultant velocity component

vn = Normal local component of wind velocity

uy = Wind velocity in the local y direction

vt = Axial(x) component of wind velocity

Projected Method - In this approach a wind pressure based on the resultant wind velocity is evaluated and then applied to the projected element areas in the direction on the wind using $\rho U^2 Cd.D.L_{Pw} / 2$

Wind Interaction Method option parameter = 1 Default method

$P_W = \rho U^2 / 2$ Wind pressure

$F_W = P_W.(Cd.D)_W.L_{Pw}$ Force in direction of wind

Where:

L_{Pw} = Projected length normal to the wind direction

$(Cd.D)_W = CDy.Dz.\sin^2(\alpha) + CDz.Dy.\cos^2(\alpha)$

CDy = Drag coefficient for local y component of flow normal to the member

CDz = Drag coefficient for local z component of flow normal to the member

α = ATAN(uy/vn)

vn = Normal local component of wind velocity

uy = Wind velocity in the local y direction

Axial Loading (Local x direction) Loading

Loading in the axial direction is based on the following

Fx = CDx.RHO.Lp.vt..vt/2

Where:

CDx = Drag coefficient for x comp flow (axial to the member)

Lp = Effective perimeter (Coating perimeter for non-circular sections)

vt = Axial(x) component of fluid particle velocity

3.2 Nodal Areas and Associated Cd's - Non-Structural Loading

Loading due to non-structural attachments can be accounted for by the use of nodal Areas. Area and their associated local direction coefficients can be assigned to specific nodes in the model. The wind interaction method is defined for each area.

Component Method - In this approach the resultant wind is resolved into local components and these are used to evaluate the forces in the local directions.The component method is most suited for representing solid entities which have an effected loaded area in both the x and z global planes (when referenced to a vertical member)

The directional properties are defined by reference to the orientation of an existing element or by reference to a local coordinate system.

The evaluation of these types of load are based on the following

Fn = CDn.ρ.A.vn.vn/2

Where:

CDn = Drag coefficient for comp flow in the direction of the local axis

A = Normal Area

RHO = Fluid mass density

vn = Component of fluid particle velocity in the local direction

Projected Method - In this approach a wind pressure based on the resultant wind velocity is evaluated and then applied to the projected area in the direction on the wind. The projected method is most suited to cases uni-directional loading such as a flat plate disk.

The normal direction of the Area is defined using global X,Y and Z coordinates.

P_W $= \rho U^2 / 2$ Wind pressure

F_W $= P_W.Cd.A_{Pw}$ Force in direction of wind

Where:

CDn = Area Drag coefficient

A_{Pw} = Projected Area = A.sinϕ

ϕ = Angle of the wind relative to the normal of the area

ρ = Fluid mass density

U = Resultant velocity component

3.3 Linearisation of Element Loads

Since the aerodynamic loading may not be linear (i.e. when velocity profiles are specified) along the element, each element is divided into segments for load evaluation. The loading is then approximated by a linear trapezoidal load distribution across each segment. The default number of divisions is 2 but this can be increased up to 10 for long elements or cases with large wind variation.

-o-

4.0 Creating Structural Models for Wind Analysis

Restrictions and/or cautionary measures when creating structural models in FS2000 for wind loading are;

- The ground must be in the X-Z plane with the Y direction positive in the direction of increasing height. This is shown in the Figures.
- When saving wind load cases ensure that different load case numbers to existing load cases are used, otherwise they will be overwritten. It is good practice to keep wind load case numbers high e.g. 50 upwards.
- FSWind creates secondary binary load files which contain trapezoidal patch loads for each element and a text load case definition file (.L"m"). The .L"m" file only contains the total shear loads and identifies that the loading is from FSWind. If a model is changed or saved in the Model Definition TASK , then all wind load cases must be regenerated since the secondary load files are purged (deleted). If the Purge Results option is not active when the model is saved, the existing load cases do not require to be regenerated. This is not recommended practice
- If results cases are purged then the wind load cases will require to be regenerated. If the model is recovered from archive, the wind load cases require to be regenerated
- If the number of elements in the model is changed the wind load cases will require to be re-generated. A warning will be given if this happens.
- The loading on tapered elements is based on the mean width.

UNITS

The units used in FS-Wind are dependent on whether the model was created using SI units or US units. The unit table shows the unit that must be used and the default values.

-o-

4.1 Units

The following table shows the units that must be used in the FS-Wind and the default values

FS2000 Model Definition	SI	US Units
Ref Height	m	ins
Current velocity	m/s	ft/s
Profiles heights	m	ins
Density	1.226 kg/m3	0.0765 Lbs/ft3
Dynamic viscosity	1.725E-5 Ns/m2	1.16E-5 Lbs/s ft
Gravity	9.81 m/s2	32.2 ft/s2
Nodal areas	m2	ft2
Member Size and Ice coatings	m	ins

-o-

5.0 Loading Properties

Gravitational Loading

Gravitational loading due to a coating thickness and a content density can be accounted for.

A coating perimeter must be specified for non-circular sections if coating weight is to be included.

For Non-Circular hollow section an internal area must be specified for contents weight to be included.

Element Coefficients & Data

By Geometric Property Code Reference

By default all element coefficients are defined in terms of their geometric property code. The may also be define on an individual element basis.

The default coefficients to be assigned by default can be re-defined by the user. These defaults are be applied to all codes.

The individual properties assigned to the property codes can be modified. Properties that are modified can be identified in the View List by the * character. Modified and non-modified properties are written to different sections of the WIN file. Non-modified properties are written to the WIN file solely to record the values being used. They are never interpreted.

The property code parameters used are:

Cd-y	Drag coefficient. in the local element y-direction.
Cd-z	Drag coefficient. in the local element z-direction.
Dy	Characteristic dimension in the local y-direction (for local z flow).
Dz	Characteristic dimension in the local z-direction (for local y flow).

For **pipe elements** the O.D. will be assigned to Dy & Dz by default and the Cd values will be assigned 0.7.

For **non- pipe sections** then default values for Cd is 2 . These defaults can be re-defined by the user. The Dy and Dz will be assigned to 2Ly(1) and 2Lz(1). This equates to the depth and width of the section for symmetrical library sections.

If the Cd-y coefficient is made negative then the coefficients for that property code will be de-activated and the coefficients will be evaluated by the active function method i.e. depth profile or Re function.

The other properties defined by geometric property code are:

- Internal Area
- Coating Perimeter

The Internal Area is used to evaluate the gravitational load of any defined contents. Therefore for hollow non-circular sections ensure that the internal area is defined.

The **Coating Perimeter** is used to evaluate the weight of coatings (ice) on non-circular sections and is also used for the evaluation axial drag in non-circular sections.

Box sections are identified if property GType parameter is a B (GTAB1 command). In such cases the Coating Perimeter is evaluated using (Dy + Dz)*2 and the Internal Area using (Dy -t*2)*(Cz-t*2) where t = ave t = CSA/Perimeter.

IMPORTANT COMMENT

When property code referenced coefficients are modified and are saved, these values are fixed. They can only be changed by subsequent re-definition and re-save. Any future modifications to the property codes in the FS2000 model will not be change the coefficients already saved. If Property Codes data is changed in

FS2000 such that the coefficients in the Aerodynamic Model Data are invalid that the values will require to be modified. The Upgrade form Prop Codes button will do this all codes within the model.

By Element Reference

Coefficients may be defined by element reference. Element reference takes precedence over property code definition. The following properties can be defined by element reference;

Contents Density

Cd-y	Drag coeff. in the local element y-dir.
Cd-z	Drag coeff. in the local element z-dir.
Cd-x	Drag coeff. in the local element x-dir (axial).

Cd by Profile Definition

If the Cd-y property code coeffcicient is a negative value the property code drag coefficients will be disabled and the coefficients will assigned using the active profile method i.e. Cd profile or Reynolds Number correlation.

Nodal Coefficients

Nodal Areas are used to attract drag loading. Three coefficients that correspond to the principle directions x, y and z may be defined for each type. Orientation of the coefficients is defined by equating them to one of the following.

For Component Interaction Method:

- Global axis
- Reference to an existing element
- Reference to a local coordinate system

For Projected Interaction method:

- Defining the normal of the Area using global coordinates

-o-

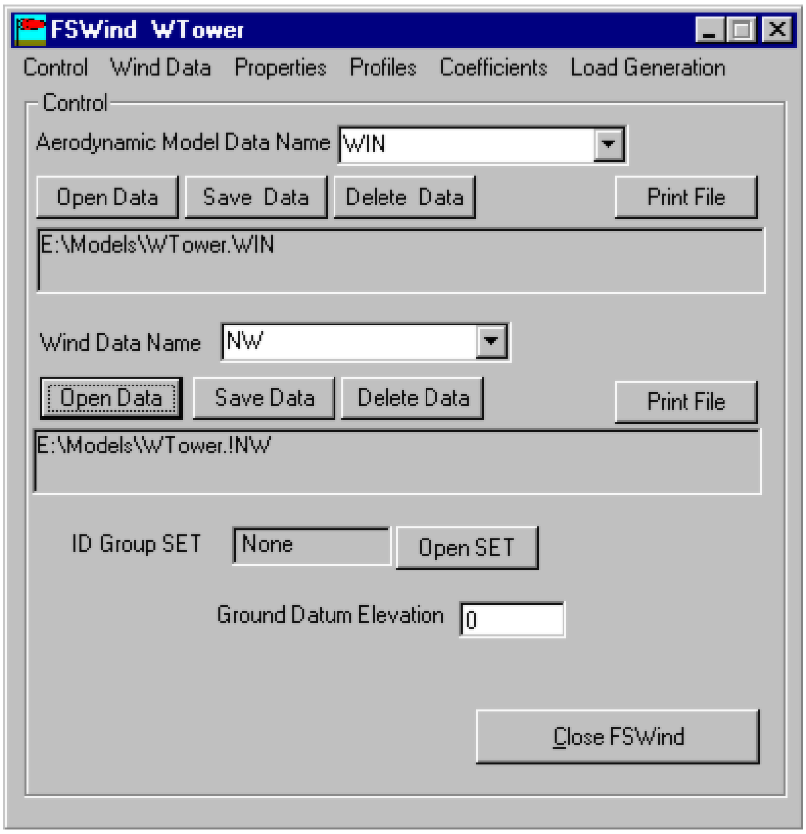
6.0 Operation

A data entry window will appear once the program is loaded. The top menu bar used to select the any of the following pages for data entry and program operation.

Control	Definition Data File management
Environment	Definition of wind loading
Properties	Basic property definition
Coefficients	Definition of load coefficients
Profiles	Defines load related profiles
Evaluation	Initiates and Controls load evaluation

-o-

6.1 Operation - Control



The unit table shows the unit that must be used and the default values.

Aerodynamic Model Data

On entry the default Aerodynamic Model Data File (.WIN) will be loaded, if one exists. The text box indicates if one is loaded. The upper **Open Data** button is used to open the user defined Hydrodynamic Model data file whose name is entered in Name box. Existing data can be selected from the drop down list. The upper **Save Data** is used to save the Aerodynamic Model Data currently in memory.

When saving the Hydrodynamic Model Data File it is always essential to start the name with **WIN** (WIN files are identified by the file extension .WIN).

If additional Hydrodynamic Model Data File are required e.g. using different profiles on the same model, then **WIN** must always form the start characters of the name of the additional data e.g WIN2. This will ensure that all definition data is archived.

Environmental Data

The lower **Open Data** button is used to open the existing Environmental Load Data file whose name is entered in Name box. Existing data can be selected from the drop down list. The lower **Save Data** is used to save environmental data currently in memory.

Environmental data is the data shown when the Environmental Tab is selected all other data is Aerodynamic Model related data.

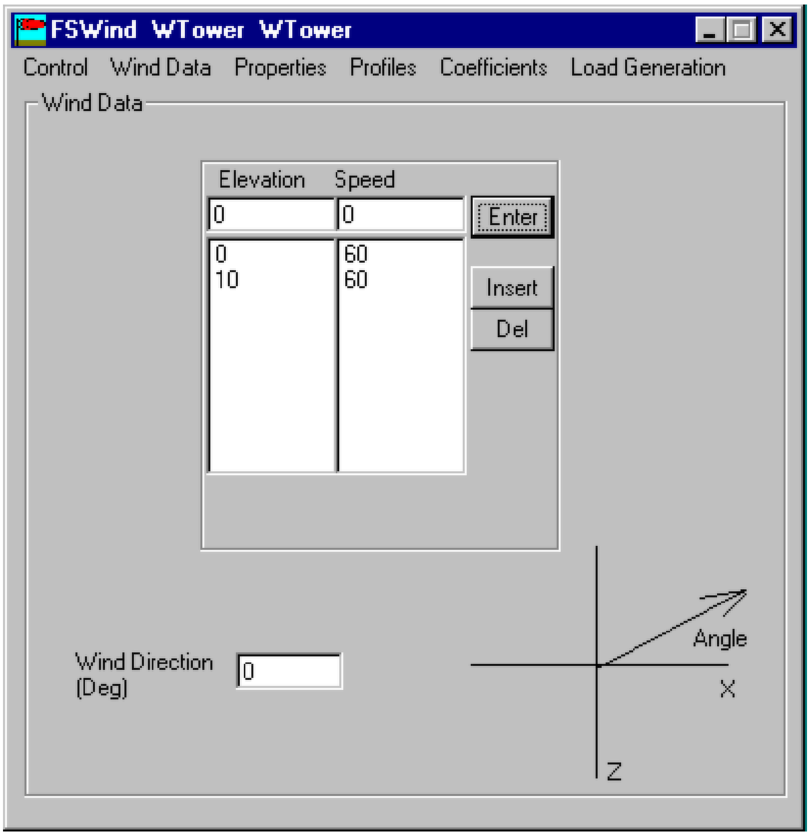
To ensure that the Environmental Load Data File is archived with the model, the name must be the model name and the first character of the extension must be a **!**. When entering a data name it is not necessary to include the **!,** it will be added by the program.

The ID Group Set **Get SET** is used to load a Group SET. Group identification can be used to select elements for data assignment.

The **Ground Datum Elevation** is used to define the position of the zero elevation wind profile datum (Ground) relative to the model co-ordinate system origin. See Figures.

-o-

6.2 Wind Data Input



The unit table shows the unit that must be used and the default values.

Wind Profile data is entered using the input boxes to the left of the **Enter** button. Use the **Enter** button to add the displayed data to the profile list. The list may be edited as required using the **Insert, Del** and **Enter** buttons.

Profile Convention

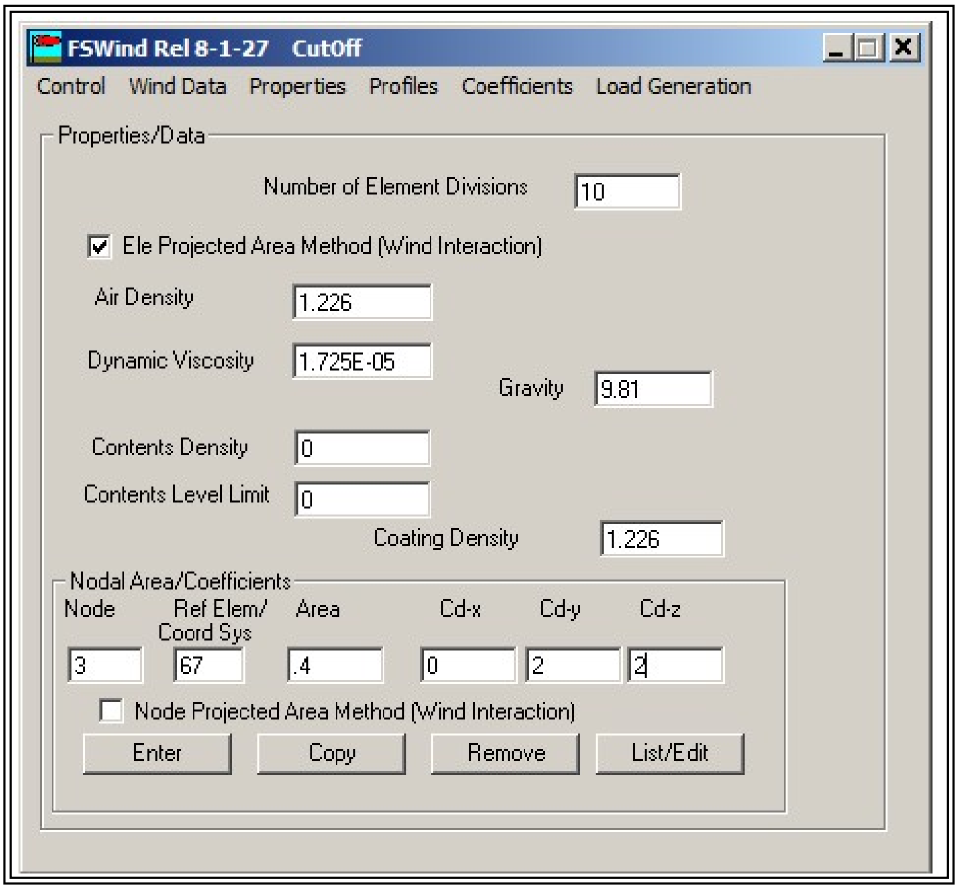
The **Elevation** is entered as positive quantity when defining velocities above ground. Always ensure that the lowest profile point is the first in the point in the profiles list i.e. the first point is towards or at ground level.

Up to 10 points may be used to define the current profile. At least two data point must be entered. Velocities at elevations above the defined profile will take on the value of the highest profile point. Wind velocity will be cut off blow the datum elevation (Ground).

Wind Direction are used to define the wind parameters. Wind direction follows the convention of that shown in the figure on the form (45 deg shown on form)

-o-

6.3 Basic Properties/Data



The unit table shows the unit that must be used and the default values.

Ele Projected Area Method This defines the member/wind interaction method 0 - Component Method 1 - Projected Area Method (Option Checked) See Section 3.0

Most of the above entities are self explanatory apart from the following, which are described in more detail.

Number of Element Divisions This option allows the user to define the number of element divisions for force evaluation. The program divides the element into a series of trapezoidal loads for load application. The larger the number of divisions the greater the accuracy but the longer the computational time. The maximum number is 10.

Contents Density This is used to specify a global contents density for all hollow elements. For non pipe section the internal area has to be defined by individual property code definition.

Contents Level The contents level limit can be used to define an upper limit above which the hollow section will be assumed to have zero density. The elevation should be defined in terms of its position relative to ground i.e. 10 would be 10 above ground.

Gravity is used to evaluate coating weight effects

Nodal Areas and Associated Cd's

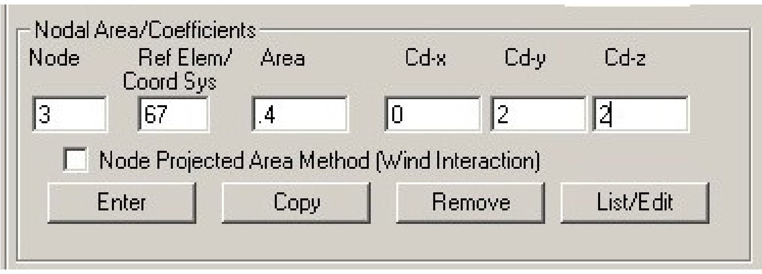
Node Projected Area Method 0 - Component Method 1 - Projected Area Method (Option Checked)

See Section 3.0 Although this is a global setting, it is possible to assign mixed nodal methods by defining the AREA as a negative value. If the Projected Area Method is not active then those with a defined negative AREA will be evaluated using the Projected Area Method.The converse is not the case. Clicking the check box changes the label display.

The Nodal Areas frame is used to define areas to be attached to nodes.

The **Node** box is used to identify the recipient node. A double mouse click will enter the last node queried in FS2000.

If the Projected Area Method is not active the input variables will be as shown as below



The **Ref Elem/Coord Sys(-)** is used to define the orientation of the attachment by equating the directions of the local drag to this entry.

-1 Aligns to the global axis

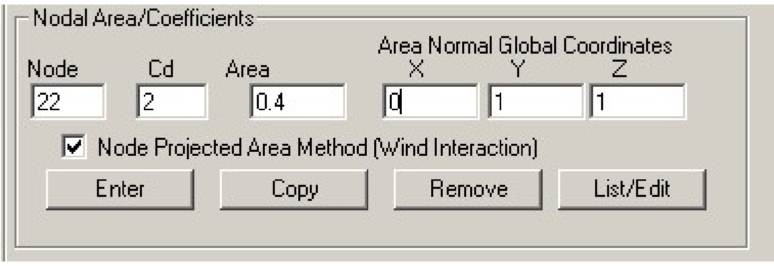
>0 Aligns to the orientation of the specified element (by element label)

<0 Aligns to the orientation of local coordinate system (e.g. -6 is CSys 6)

The **Area** box is used to define the magnitude of the attachment type.

The **Cd-x**, **Cd-y** and **Cd-z** boxes are used to define the local drag coefficients relative to the associated coordinate system.

If the Projected Area Method is active the input variables will be as shown as below.



The **Cd** box define the Cd in a direction normal to the **Area**

The Area is the area to be attached to the node.

The **X**, **Y** and **Z** define the global coordinates of the normal of the Area e.g. 0, 0, 1 would define area normal to the global z axis

The **Enter** button is used to assign the current data setting to the node. More than one entry can be assigned to a node so take care not to duplicate.

The **Copy** button is used to assign the current setting to a range of nodes that can be identified by label or group attribute.

The **Remove** button is used to delete the node attachment to a range of nodes that can be identified by label or group attribute.

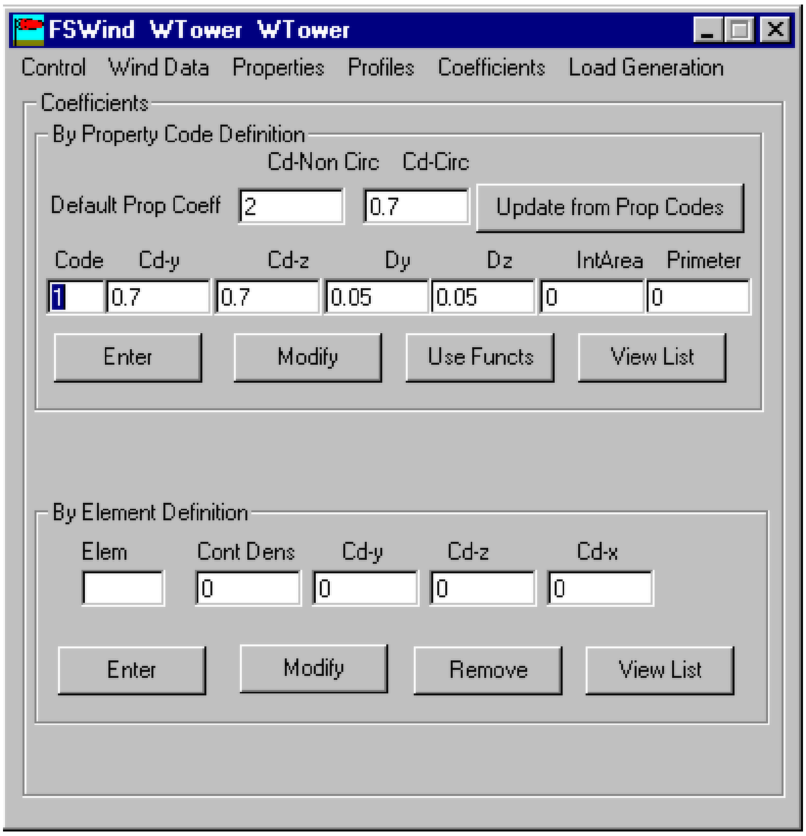
The **List/Edit** button is used to show a list of all the nodal attachments. The list has Edit and Delete options. When the Edit is clicked the entry will appear in the input box.

-o-

6.4 Coefficient Definition

The Coefficients input form is divided into two basic sections.

- The By Property Code Definition section is used to enter or edit Aerodynamic Coefficients referenced by geometric property code.
- The By Element Definition is used to enter or edit Aerodynamic Coefficients referenced by element label. Data defined by element definition always takes precedence.



The unit table shows the unit that must be used and the default values.

By Property Code Definition

IMPORTANT COMMENT

When specific property code referenced coefficients are modified and are saved, these values are fixed. They can only be changed by subsequent re-definition and re-save. Any future modifications to the property code in the FS2000 model will not be change the coefficients already saved. If Property Code data is changed in FS2000 such that the coefficients in the Aerodynamic Model Data are invalid that the values will require to be modified. The Upgrade form Prop Codes button will do this all codes within the model. Note that this only applies to individual property codes that have been modified. Property codes which have been modified are indicted by a * in the **View List**.

If **Dy** or **Dz** is specified as zero (the default) then these will always be assigned the default values from model property code. The **View List** will display the values used.

The **Default Prop Coefficients** are the default values that will be applied to non-modified circular and non-circular sections.

When the **Code** number is entered the data may be edited. See Section 3 and Section 5 for more details.

The **Enter** button is used the enter the data shown in the boxes.

The **Modify** button is used to copy the displayed entries to other property codes. When selected the user is required to input a code number range.

The **Update from Prop Codes** button is used to initialise the whole list based on the current property codes. This would be used in cases where the model property codes have been changed for codes that are currently saved in the definition list. **Note**: Individual entries can be updated by editing the WIN i.e. delete the appropriate PCOF line and then re-start FSWind.

If the **Cd-y** coefficient is made negative then the coefficients for that property code will be de-activated and the coefficients will be evaluated by the active profile method i.e. CD height profile or Reynolds No correlation. The **Use Functs** button can be used to de-activate/activate the code definition using a code number range.

The **View List** button is used to display the list of all the Code data. The list has the following button controls.

Edit This button enables selected entries to be edited

Update This button is used to update the list following data changes

By Element Definition

Zero entries will be ignored.

The **Enter** button is used the enter the displayed data shown in the boxes.

The **Modify** button is used to copy the displayed entries to other elements. When selected the user is required to input either an element label range or an identification element group.

The **Remove** button is used to remove all entries on selected elements (by range or group)

The **View List** button is used to display the list of all the element data. The list has the following button controls.

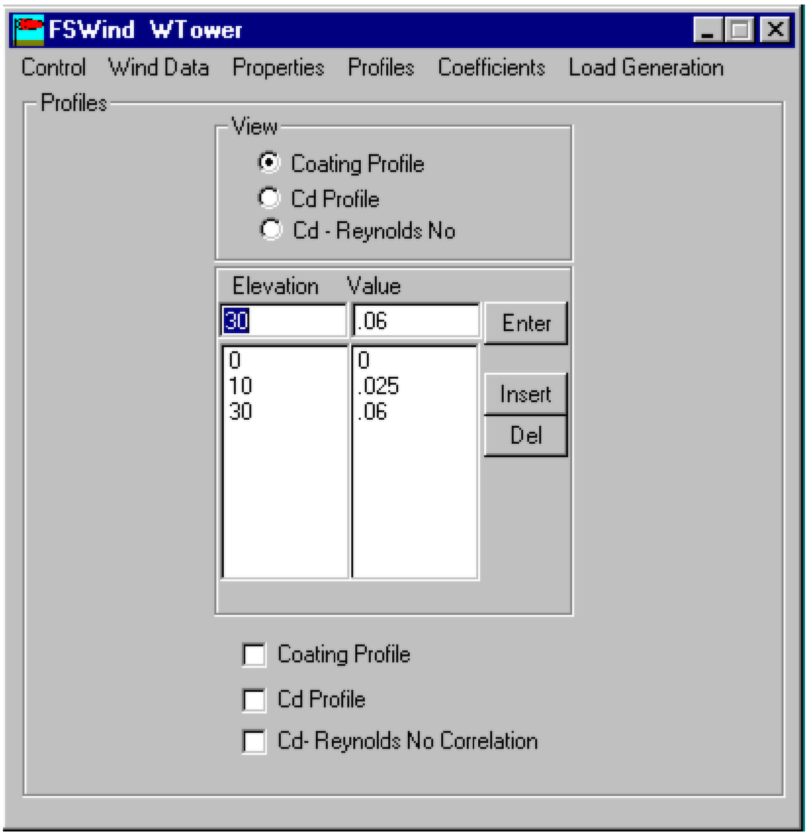
Edit This button enables selected entries to be edited

Update This button is used to update the list following data changes

-o-

6.5 Profile Definition

The Profiles Tab form is used to define Coating-Height profiles, Cd-Height profiles and Reynolds No-Cd profiles.



The unit table shows the unit that must be used and the default values.

The **View** option is used to select the profile to be displayed in the profile list.

The **Check** boxes at the bottom of the screen are used to select the profiles to be used for load evaluation. Only one of the CD profiles can be active at one time.

Profile data is entered using the input boxes to the left of the **Enter** button. Use the **Enter** button to add the displayed data to the profile list. The list may be edited as required using the **Insert, Del** and **Enter** buttons.

Profile Convention

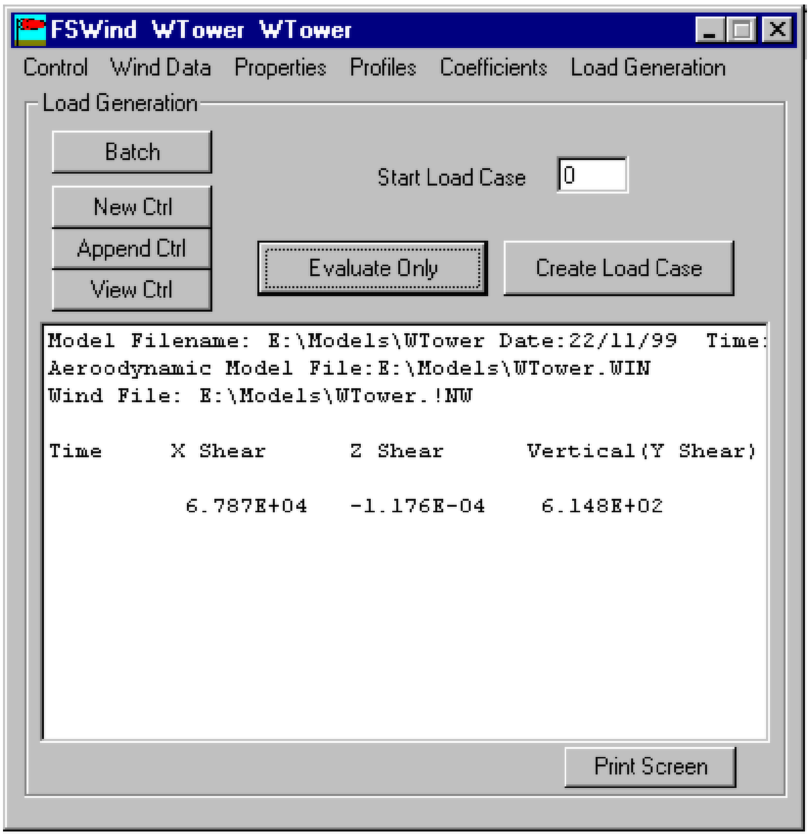
The **Elevation** is entered as +ve quantity when defining values above the datum elevation (ground). Always ensure that the lowest profile point is the first in the point in the profiles list i.e. the first point is towards or at ground level.

Up to 10 points may be used to define a profile. At least two data point must be entered. Values at elevations above or the below the defined profile will take on the value of the highest or lowest profile point value. Therefore ensure that the ground value is zero, unless the coating load is to be applied below the ground elevation.

-o-

6.6 Load Evaluation

This Tab form is used to initiate load evaluation.



Start Load Case This box is used to define the Load Case number.

If the **Echo Forces** is active, a file (**<ModelName>.WLD'Load Case No')** containing the element local distributed loads will be created. The loads are in the ED command line format.

The **Evaluate Only** button is used to evaluate the shear loads on the structure without creating a load case. The shear loads will be displayed in output list.

The **Create Load Cases** is used to generate a load case using the **Start Load Case** as the load case number.

The following buttons are used for batch operation. For a more detailed description of batch control files see Command Line Operation.

The **Batch** button id used to append the FSWind command line to the **<ModelName>.BRM** batch file.

The **New Ctrl** button is used to create a new windload batch control file **<ModelName>.BCA** using the current settings.

The **Append Ctrl** button s used to append the current settings to the existing windload batch control file **<ModelName>.BCA.**

The **View Ctrl** button is used to view the existing windload batch control file **<ModelName>.BCA.**

-o-

7.0 Command Line (Batch) Operation

The following describes the command line syntax that may be used to activate FSWind.

To use the command line activation it will be necessary for a ***WINFILE*** to exist and an additional ***Batch Control File*** (Wind Control Files) called "modelname".BC**WIN** which is used to control the specific load case generation, to also exist. **WIN** is the default WIN file extension name. More that one file may be used e.g. WIN2.

The ***WINFILE*** and any associated ***Batch Control File*** must be linked by name e.g. **WIN2** matches BC **WIN2** and both must exist.

The command line instruction to run FSWind is

WINDLOAD *WINFILE*

WINFILE is used to define the Aerodynamic Model Data File to be loaded. Only the model file extension should be used e,g. **WIN2**. Note that the extension name must start with the 3 characters **WIN**

If MODEL is used for ***WINFILE*** then the <modelname>.WIN data will be loaded (legacy command).

Note that the Batch button in the FSWind form uses the current WIN file extension name when the WINDLOAD command is entered in the batch file.

The **WINDLOAD** command will start the FSWind, load the **Aerodynamic Model Data File** and then look for the **Batch Control File** which is described below.

Wind Control File (Batch Control File)

The control file "modelname".BCWIN must exist in the same directory as the model. This text file controls the operation of FSWind when running in batch mode. The format for this file is given below.

NU	number of wind file to be read
ENVNAME	environmental file name e.g. WEXP.!NW, better to use MODEL.!NW
START	wind load case start number

The latter 2 entries require to entered NU times.

The **ENVNAME** can be any file specification and need not be related to the model. If **ENVNAME** is entered as MODEL, then the default model name will be assumed (this makes the command portable i.e. model independent - which is highly desirable when copying models i.e. Save-as).

A typical <ModelName>.BCWIN is below. (MODEL is used to infer the ModelName)

```
2
MODEL.!NW
10
MODEL.!SW
12
```

The **New Ctrl** and **Append Ctrl** buttons in the Load Evaluation Tab form are used to transfer the current settings to the control file.

-o-

8.0 Data File Formats

This section describes the command line instructions used by FS-Wind for its definition files.

Note that earlier versions of FS2000 used a fixed data file structure. These can still be read by FS2000 and will be converted to the latter format if re-saved.

The defaults for all command arguments is zero.

The Aerodynamic Model Data File

NDV, *Number*

Defines the number of element divisions for force evaluation. Range is 2 to 10

WINT, *Value*

Defines element wind interaction method. 0 - Component or 1 - Projected

WINTN, *Value*

Defines nodal area wind interaction method. 0 - Component or 1 - Projected

GDAT, *Coordinate*

Coordinate of ground relative to model origin

GRAV, *Value*

Gravitational acceleration - always 9.81 unless units are changed from S.I

APROP*, Density, Viscosity*

Air properties

ACONT, *Contents Density, Elevation Limit*

Properties of global contents

CODEN, *Coating (Ice) Density*

Density (in-air) of coating (ice)

DEFWC, *Cd Circ, Cd Non-Circ*

Default coefficients for property code assignment

PCOF*, Code, Cd-y, Cd-z, Dy, Dz*

Coefficients assigned to specific geometric property codes.

If Dy=0 or Dz=0 then model property code values will always be used.

ECOF*, Elem, Contents Density, Cd-y, Cd-z*

Coefficients assigned to specific elements

NCI, *Code, Internal Area, Coating Area*

Additional properties for non-circular sections assigned to geometric property table

NA, *Node, RefElem/CordSys, Area, Cd-x, Cd-y, Cd-z, WInt* If WInt=0

NA, *Node, Cd, Area, X, Y Z, WInt* If WInt=1

Definition of nodal areas etc.

CDWPR, *Elevation, Cd*

Definition of a Cd profile point. Data point must be in order,

COPR, *Elevation, Coating Thickness*

Definition of a caoting profile point. Data point must be in order,

REPRW, *Re, Cd*

Definition of Cd as a function Reynolds Number. Data point must be in order.

Environmental Load Data Files

WIDIR, *Current direction*

Current direction

WI, *Elevation, Value*

Defined wind as a function of elevation above ground datum. Data points must be in order

-o-

9.0 Archiving Wind Related Data

The archive facility of FS2000 will automatically archive the following definition files which are the default naming conventions when using the program..

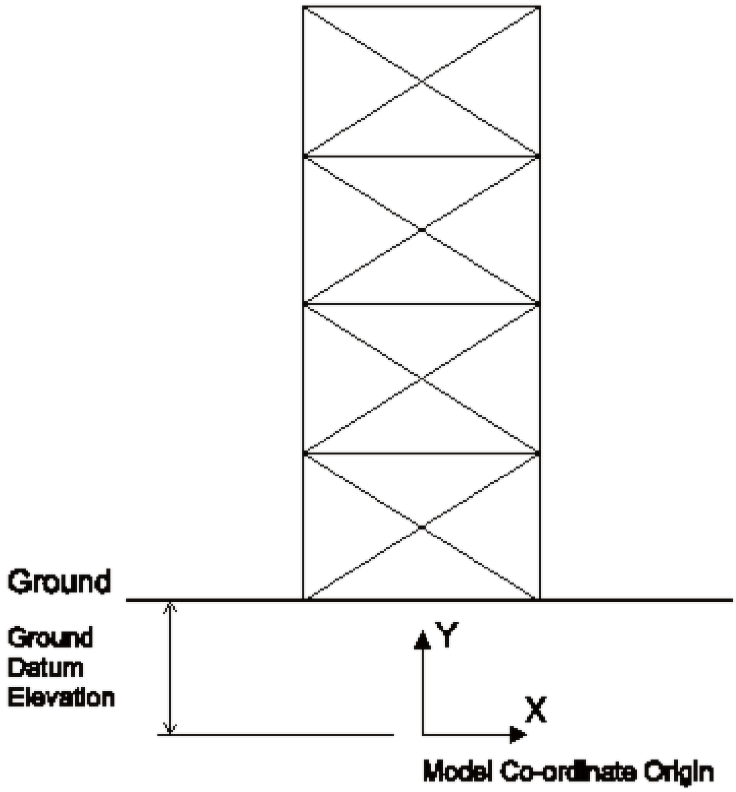
Aerodynamic Mode Data Files i.e. the **.WIN** files with the same file name as the model.

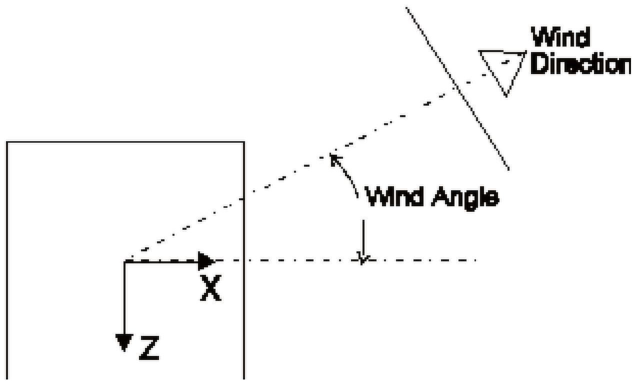
Environmental Load Files with the same file name as the model and the first character of the extension is a **'!'** character.

If additional Aerodynamic Mode Data files are required to be used use **<ModelName>.WIN*** since this type of file will also be archived.

-o-

10.0 Figures





-o-