GH Bladed

Version 3.51

User Manual

Document No	282/BR/010
Classification	Commercial in Confidence
Date	June 2003

Author:

E A Bossanyi

Checked by:

D Witcher

Approved by:

D C Quarton

DISCLAIMER

Acceptance of this document by the client is on the basis that Garrad Hassan and Partners Limited are not in any way to be held responsible for the application or use made of the findings of the results from the analysis and that such responsibility remains with the client.

Key To Document Classification

Strictly Confidential	:	Recipients only
Private and Confidential	:	For disclosure to individuals directly concerned within the recipient's organisation
Commercial in Confidence	:	Not to be disclosed outside the recipient's organisation
GHP only	:	Not to be disclosed to non GHP staff
Client's Discretion	:	Distribution at the discretion of the client subject to contractual agreement
Published	:	Available to the general public

CONTENTS

1. INTRODUCTION

Bladed is an integrated software package for wind turbine performance and loading calculations.

The ***Bladed*** theory manual gives full background information and details of the calculation methods used. This user manual complements the theory manual by describing the operation of the user interface. Much of this information is also available in the on-line Help facility.

1.1 Installation

Bladed is designed to run on PC compatible computers under Windows 95, NT, 2000 or later versions. A Pentium computer with at least 8 MB RAM is recommended.

Bladed is supplied on CD and is installed by means of a setup program. The setup procedure suggests an installation directory for the software, but allows the user to change this if desired. A directory on the local hard disk is recommended.

Except for demonstration and educational versions, a software key or dongle is provided. This should be inserted into the parallel printer port. A network dongle can also be supplied on request. The setup procedure can install the appropriate device driver and network licence manager.

The setup procedure also allows the installation of the Acrobat Reader if desired, which is used to view the User and Theory Manuals on screen. It also allows the installation of the Matlab runtime libraries which are used by the optional Linearisation module[1.2].

The preferred screen resolution is at least 1024 x 768 (small fonts), although a resolution of 800 x 600 is acceptable.

1.2 Optional modules

The ***Bladed*** package consists of a number of modules, for which licences are separately available. The following modules are available:

Module	**Capabilities**
Main module*	Steady state calculations⁺, Saving project files, Writing reports, Graphics
Simulation module*⁺	Modal analysis, Simulations, Turbulent wind, Waves, Standard post-processing
Batch module*	Running calculations in batch mode
Linearisation module	Generation of linearised turbine models
Advanced post-processing	Automatic loads reporting, batch plotting

*Available in the educational version ⁺Restrictions apply in the educational version

1.3 Support

Bladed is supplied with a one-year maintenance and support agreement, which can be renewed for further periods. This support includes a 'hot-line' help service by telephone, fax or e-mail:

Telephone:	+44 (0)117 972 9900
Fax:	+44 (0)117 972 9901
E-mail	bladed@bristol.garradhassan.co.uk

It is Garrad Hassan policy to work with clients to respond to their needs so that the software can be constantly improved. As with any software of this complexity, a total absence of bugs cannot be guaranteed, and any reports of bugs, along with comments or suggestions on any aspect of the software, are welcomed. The maintenance and support agreement includes free provision of any revisions or upgrades of the software during the period of the agreement.

Modifications to the software to meet the individual needs of specific clients can be made be arrangement. Such work will be charged for at commercial rates.

1.4 Documentation

Much of the information contained in this manual is also available in the on-line help facility.

There is also a ***Bladed*** Theory Manual which explains in more detail the theory behind the calculation methods used.

Section 2 of this User Manual gives an overview of how ***Bladed*** can be used. The later sections then provide more detail on each function. Sections 3 to 5 explain in detail how to set up a model of a turbine. Section 6 covers the specification of the wind field, and the sea state for offshore turbines. Section 7 then explains how to set up and run wind turbine calculations. The post-processing, graphics and reporting facilities are described in Sections 8, 9 and 10 respectively.

1.5 Acknowledgements

Bladed was developed with assistance from the Commission of the European Communities under the JOULE II programme, project no. JOU2-CT92-0198.

2. USING *Bladed*

This chapter gives an overview of the ways in which ***Bladed*** can be used. It covers the following topics:

- General description and layout of the user interface
- Entering data
- Using project files
- Performing calculations
- Viewing results
- Compiling reports

More detailed descriptions follow in later chapters.

2.1 General description and layout of the user interface

On starting ***Bladed***, the main Toolbar appears together with the Calculations screen. The Toolbar consists of a set of graphical icons and a number of pull-down menus. The Calculations screen allows the user to select, define and execute a particular calculation.

There are various ways of opening further screens which allow the user to define the characteristics of the various parts of the wind turbine, as well as the characteristics of the wind and various parameters which control the execution of calculations. There is also a graphics facility for viewing results.

2.1.1 Main toolbar - pull-down menus

The pull-down menus may be used as follows:

File: use this menu to create, open and save project files, and to import modules from other project files.

A project file (.prj) contains wind turbine information and/or parameters defining calculations. The project file header information may be edited, and a security password entered to prevent the file being modified by unauthorised users.

Using the file type selector on the File Open dialogue box, it is also possible to open a project backup file (.prx), or a file containing all the details relating to a calculation previously carried out (.$pj). This is useful for re-running the calculation with or without modifications.

Use the **Import** facility to import individual modules from other project or calculation details files into the currently active project.

Specify: this menu allows the user to move directly to a particular screen for specifying any part of the wind turbine, or any calculation or load case. These screens are also accessible via the Toolbar Icons described below.

Calculation: this menu allows particular calculations to be carried out. See also the Calculation icon on the main toolbar described below. It also allows unwanted calculation results to be deleted.

Batch: this is used to control batch[7.2.7] processing of multiple runs.

Reports: This menu also offers the possibility to write a project report or a calculation report, to append graphs to a report, and to edit or print existing reports. It also gives a choice of report format, which may be ASCII or Microsoft Word.

Tools: **Copy results** allows calculation results to be copied from one location to another. In doing so, the results can be converted between binary and ascii formats if required. There is also a facility to **Delete results. Compare Projects** allows two project and/or calculation details files to be compared, and can generate a detailed report of the differences. There is also a facility for configuring the current printer, and to specify user preferences for certain option settings[2.7].

Windows: switches between any of the currently open windows, and gives access to the turbine summary information window.

Help: activates the on-line Help facility, which contains detailed information on ***Bladed*** and how to use it. It also gives on-line access to the ***Bladed*** User and Theory manuals, and to a facility for upgrading the dongle or security device by entering an appropriate password.

2.1.2 Toolbar icons

Beneath the pull-down menus on the toolbar are the Toolbar Icons. Clicking on any one of these opens up the corresponding screen, as follows:

- Blades:[3] to define the blade properties.
- Aerofoil sections:[3.6] to access a database of aerofoil section data.
- Rotor:[4.1] to define the properties of the rotor and overall turbine configuration.
- Tower:[4.4] to define the tower properties.
- Power Train:[4.5] to define the drive train, generator, energy losses and electrical network.
- Nacelle:[4.11] to define the nacelle geometry and mass.
- Control:[5] to define both power production and supervisory control systems.
- The wind input:[6.1] to define the wind speed and direction including spatial and temporal variations.
- Waves and currents:[6.12,6.13] to define the sea state for offshore turbines.
- Modal analysis:[7.1] to specify the modal analysis of blade and tower vibrations.
- Calculations:[7.2] to select, specify and execute any particular calculation.
- Data View:[9] to view graphs or generate tables of results.
- Analyse:[8] to specify post-processing of results.

2.1.3 The calculation window

The **Calculations** window lists all the available calculations, any one of which may be selected by clicking on it with the mouse. Next to each calculation is an indicator light. If this is green, the calculation may be performed. A red light indicates that no data is available to perform the selected calculation, while a yellow light indicates that some of the required data has been defined but not all.

Below the calculations is a small window showing all the data modules which need to be defined in order for the selected calculation to be performed. Double-clicking on one of these data modules immediately opens up the relevant screen in which the data for that module is defined. It is therefore a simple matter, having selected the calculation to be performed, to work through all required data modules which are still undefined and assign the relevant data.

2.1.4 Sequence of operations

Before any wind turbine calculations can be done, it is necessary to specify the principal characteristics of the blades and rotor. Open the relevant screens by one of the means described above, i.e. from the **Specify** pull-down menu, from the relevant Toolbar icon, or from the **Calculations** screen with one of the wind turbine calculations selected.

In order to be able to define the aerodynamic characteristics of the blades, it is necessary to enter or import the relevant aerofoil datasets into the aerofoils database if it is not already present, using the **Aerofoil** icon on the toolbar, or from the **Specify** pull-down menu.

It is also necessary to define the following modules:

- Physical constants (although standard values will have been defined by default)
- Aerodynamics control
- Calculation definition depending on which calculation is selected (further parameters specific to the selected calculation)

These are all on the Calculation Parameters screen, accessible from the **Calculations** screen or from the **Specify** pull-down menu on the toolbar.

Having defined this fundamental data, it should then be possible to perform the following steady state wind turbine calculations at a fixed rotational speed and pitch angle defined on the Calculation definition screen specific to each calculation:

- Aerodynamic information (to examine the aerodynamics at each blade station, such as lift and drag, inflow, tip loss etc. at a specified wind speed)
- Performance coefficients (dimensionless power, torque and thrust coefficients as a function of tip speed ratio)
- Power curve (at specified fixed rotor speed and pitch angle).

These calculations are quick and the results can instantly be examined graphically using the **Data View** icon on the toolbar. From the power curve, the annual energy yield can also be calculated. This is one of the post-processing calculations, and again the results can be seen using **Data View**.

The **Outputs** button on the **Calculations** screen allows the user to specify in some detail which loads and other outputs are desired. Once this is done, the Steady-state operational and parked loads calculations can also be carried out (at specified fixed rotor speed and pitch angle), and **Data View** used to see the results.

At the preliminary design stage, various turbine parameters can be adjusted and all these calculations can be repeated rapidly until satisfactory results are obtained.

As further data is defined, more complex calculations can be carried out. For example, by defining suitable combinations of drive train, generator and control characteristics, it becomes possible to calculate the steady power curve and steady operational loads with the rotor speed and/or pitch angle varying with wind speed as appropriate.

After defining the numbers of vibrational modes on the **Modal** screen, along with the necessary mass and stiffness information as indicated on the **Calculations** screen, it becomes possible to perform a Modal analysis calculation. Blade and tower bending can then be taken into account in the calculations.

Simulations can also be performed once the Simulation control and Time varying wind modules are defined. As always, the minimum data required is always indicated on the **Calculations** screen, making it easy to ensure that the necessary data is defined for each calculation to be done.

Note also the **Show options** button at the bottom of the **Calculations** screen. This provides a rapid means of switching particular features off or on for a particular calculation. For example, provided the energy losses module has been defined, the energy losses can be switched off or on very easily using this feature, for all calculations where energy losses are actually relevant.

2.2 Entering data

In each window, data is entered by typing in the required information in the fields provided. Where there is a choice of alternative options, selection buttons or pull-down selectors are provided. In some cases check boxes are provided for enabling or disabling particular items (the item is selected when a tick appears in the box).

Where numerical values are entered, except in the case of dimensionless numbers, the units in which the data is to be specified are shown. Double-clicking on the entry brings up a units conversion box, allowing the user to enter values in a choice of different units.

Once data in a window has been edited, it is necessary to assign the changes by clicking the **OK** button. Alternatively, any changes made since the data was last assigned may be reversed by clicking the **Cancel** button.

2.3 Using project files

A project file stores all the currently defined information relating to the wind turbine, the wind field information, and calculation parameters. Click **OK** on any open windows to ensure that the data is assigned before saving to a project file.

A project report may be generated from the **Reports** pull-down menu on the main toolbar.

2.4 Performing calculations

Calculations may be initiated either from the **Calculations** pull-down menu on the toolbar, or from the **Calculations** window by clicking the **Run Now** button. For some of the steady state calculations and all the simulations, the **Calculations** window also gives access to facilities for switching various calculation options on or off, and for specifying the calculation outputs required. Post-processing calculations may also be initiated using the **Execute** button in the post-processing window obtained by clicking the **Analyse** icon on the toolbar.

Long calculations may stacked up using the **Run in Batch** button in the **Calculations** window. The whole batch of calculations may then be started so that they run one after the other, for example overnight. This is done using the **Batch** menu item on the toolbar.

Except for some auxiliary calculations, the user will be asked to specify where the calculation outputs are to be sent. A dialogue box allows the drive and directory to be selected, and a run name must also be specified. This is used to identify all the output files produced by that calculation. A new output directory may be created by editing the directory name.

While the calculation is running, a window appears which displays a progress bar showing how far the calculation has progressed, together with any warnings which may arise during the calculation. An ‘Abort’ button allows the user to stop a calculation which is in progress. Any part-written output files which have already been produced should be available. This means that a long simulation can be stopped without losing the results which have so far been produced.

2.5 Viewing results

After a calculation has been completed, the results can be viewed by clicking the **Data View** icon on the toolbar, and selecting the desired data for each channel. Up to six channels may be plotted on one graph. The outputs from any calculation just completed will normally be selected by default. Otherwise, the dialogue box allows any drive and directory to be selected. If the results of several calculations are available within the directory, a pull-down selector allows the user to choose the run name which was specified when the calculation was initiated.

The output files available from that calculation are shown in the Data Group window. Selecting one of these displays a contents list for the file from which the user chooses the variable of interest. Click **OK** to assign the data to the selected graph channel.

Often there is just one independent variable, for example Time in the case of dynamic simulations. Sometimes there is a choice of independent variables: for example a blade bending moment from a simulation could be plotted either against time or radius. Double-click to change the choice of independent variable.

If two independent variables are defined, a three-dimensional graph will result. Three-dimensional plots are most useful for small datasets.

The **View Messages** button displays any warning or error messages generated during the run. In a few cases, some additional information is available by clicking the **Further Info** button. Other buttons allow whole runs or individual data groups to be deleted.

2.6 Compiling reports

If Microsoft Word is installed, it is possible to generate neatly formatted project and calculation reports in Word format. It is also possible to insert calculation results into these reports, either as tables or as graphs. See also User Preferences[2.7].

If Word is not available, change the report format to ASCII using the **Reports** pull-down menu. This will cause reports to be generated in tab-delimited ASCII format, suitable for reading into other packages if required. This option does not allow graphs to be inserted into the report.

Reports are generated from the **Reports** pull-down menu on the main toolbar. There are two types of reports:

- **Project reports**: these may contain any currently assigned modules defining the wind turbine itself, as well as calculation parameters and wind field details if required.

- **Calculation reports**: these may be generated for any calculation which has been carried out, or which is awaiting execution in the Batch queue. A dialogue box allows the desired calculation to be selected Calculation reports may include any turbine details which are relevant to the calculation, as well as details of the calculation itself and the wind field used if applicable. They may be generated as stand-alone reports, or appended to another report such as a project report.

When reports are generated, they may be appended to existing report files if desired. Thus it is possible to create a project report, and subsequently append calculation reports to it whenever calculations are done, and also append calculation results either as graphs or tables.

2.7 User preferences

Select **Tools** and **Preferences** from the main toolbar menu, and select desired options as follows:

Make backups of project files: to make a backup file (.prx) whenever a project file (.prj) is opened.
Warn when starting calculation: to display a warning when a shelled calculation is about to start.
Start batch jobs in foreground: depending on the operating system, jobs may run faster when running in the foreground.
Use binary format for calculation results: binary output is faster and results in smaller files.
Report format: Select ASCII or WORD.

Insert graphs as links: to append graphs to WORD reports as links to external metafiles. If this option is not selected, graphs will be fully imported into the WORD document.
Network Dongle: This option must be selected if you wish to use a network dongle, otherwise it must be de-selected. If you have a network dongle you can change the way in which the network is searched by clicking **Change configuration**.

2.8 Context-sensitive help

For context-sensitive help, press the F1 key at any time.

3. DEFINING THE TURBINE BLADES

This section describes how to construct a model of the wind turbine blades, including geometrical, mass and stiffness properties as well as the aerodynamic characteristics of the aerofoil sections.

Click the **Blades** icon on the toolbar to open the **Blade Properties** screen.

Before any calculations can be carried out, the user must define the geometry[3.2] and aerodynamic characteristics[3.6] of the rotor blades.

If any dynamic calculations are required, check the **Mass** checkbox to allow the blade mass distribution[3.3] to be defined.

If blade flexibility[7.1] is to be modelled, also check the **Stiffness** checkbox to allow the blade stiffness distribution[3.4] to be entered.

If flap twist coupling is to be modelled, also check the **Flap-Twist Coupling** checkbox to allow torsional rigidity, mass moment of inertia and principal neutral axis for bending to be entered. This option is only enabled if both the **Mass** and **Stiffness** checkboxes have been checked.

All these blade properties are defined at a number of stations[3.1] along the blade, although additional point masses[3.3.2] may also be defined anywhere on the blade by clicking the **Point masses** button.

The **Graph** button controls the graphical display of blade properties. The graph is updated as soon as any new values are assigned, giving an instant indication if faulty values are entered. Controls are provided to change the format of stiffness graphs, and to print the graph or save it to a metafile, which can subsequently be incorporated into a report.

Click the **Add** button to add a new blade station. Stations are automatically sorted by radial position. To highlight a station, click on the station number, or double-click on a graph point. Click **Delete** to remove a highlighted station.

To enter or edit data, highlight the data item by clicking on it, or by moving to it using the arrow keys and then pressing the Return key. Press Escape to restore the previous value.

Highlight a block of cells by dragging the mouse. These cells may be copied to the clipboard, or the contents of the clipboard may be pasted in. In this way, data from a spreadsheet or a tab-delimited ASCII file may be directly pasted in. Click **Undo** to reverse a paste operation.

Note that the Moving/Fixed and Foil Section entries may not be pasted in.

Click on Moving/Fixed to specify which parts of the blade are pitch or aileron controlled. Click on Foil Section to select an aerofoil section[3.6] or to define a new section.

If a sharp discontinuity[3.1] in blade characteristics is required, split a blade station into an inboard and an outboard station at the same radial position. This is especially useful to mark the discontinuity between a fixed and a pitchable or an aileron-controlled part of the blade.

Use the **Split** and **Join** buttons to split or re-join highlighted blade stations. The first and last stations may not be split.

3.1 Choosing blade stations

Blade stations are the points along the blade at which blade geometry, aerofoil, mass and stiffness data are defined. They are also the points at which blade loads will be calculated, if desired - see Defining outputs[7.2.4].

The blade stations must include the root and tip stations. Blade station positions are measured from the blade root, which is the point at which the blade is attached to the blade root section[4.2]. Thus the first blade station must be given a radial position of zero, and the radial position of the last station defines the length of the blade. This will be less than the rotor radius if a blade root section is defined.

A reasonably even spacing between blade stations is recommended. Try to define a blade station close to or preferably at the same position as any point mass which may be required - see Blade mass distribution[3.3].

Discontinuities
The blade may have sharp discontinuities in the mass distribution, the stiffness distribution, and the aerofoil section. There will also be a discontinuity where a blade changes from fixed to pitchable, or at the start or end of any aileron or airbrake section.

To specify such a discontinuity, highlight the blade station at the radial position where the discontinuity lies, and click the **Split** button to split the blade station into an inboard and an outboard station at the same radial position. The data may then be specified differently on either side of the discontinuity.

A split station may be re-joined if necessary to remove the discontinuity, by clicking the **Join** button.

3.2 Blade geometry

The blade geometry is defined at each blade station[3.1] clicking on the data item to be defined or changed. The following data is required at each station:

- **Distance from root**: the distance from the blade root to the current blade station. It must be zero for the first station.

- **Chord**: the distance from the leading edge to the trailing edge, i.e. along the chord line.

- **Twist**: the angle between the chord line and the blade reference plane[3.2.1]. More positive values of twist and set angle push the leading edge further upwind. The blade is assumed to be twisted about the quarter-chord point.

- **Thickness**: the thickness of the blade as a percentage of the chord at that station.

- **Pitch axis**: the distance from the leading edge to the blade pitch axis as a percentage of the blade chord at that station, measured from the leading edge towards the trailing edge. For a non-pitching blade, an arbitrary axis may be defined. Pitching moments will be calculated about this axis.

- **Pre-Bend:** the distance that the blade bent away from the blade reference plane[3.2.1]. Positive pre-bend is into the wind, ie an upwind turbine usually has positive pre-bend, in order to increase tower clearance.

- **Foil section**: an index number defining the aerofoil section[3.6] at that station

- **Moving/Fixed**: differentiates between a fixed part of the blade and a part which is movable to achieve aerodynamic regulation or braking, either by bodily changing the pitch of that part of the blade, or by deploying an aileron, flap or other aerodynamic control[4.1] surfaces.

3.2.1 The blade reference plane

This is an arbitrary plane, containing the blade pitch axis, relative to which the twist angle and pre-bend are measured.

3.3 Blade mass distribution

For dynamic calculations it is necessary to define the mass of the rotor blades. Click the **Mass** checkbox **off** if static calculations are desired and the mass data has not been defined.

3.3.1 Distributed mass

For each blade station[3.1], enter:

- the **Centre of mass** position as a percentage of chord measured backwards from the leading edge, and
- the **Mass per unit length** at the blade station

Click the **Additional Mass** tab to specify any point masses[3.3.2], vibration dampers[3.3.3], or ice build-up[3.3.4] on the blades.

3.3.2 Point masses

If additional point masses are required at specific locations along the blade, click the **Additional Mass** tab and enter the point masses in the table. Click the **Add** button to add each point mass, and then enter the data required by clicking on the appropriate entry. Note that masses are automatically sorted by radial position. To remove a mass, highlight it by clicking on its number, and click **Delete**.

For each mass, the following data is required:

- **Mass**: The Mass required.

- **Distance from root**: The radial position of the mass measured from the blade root.
- **Chordwise position**: The chordwise position of the mass, measured backwards from the leading edge as a percentage of the chord at that point.

3.3.3 Vibration dampers

Particularly when operating in stall, the aerodynamic damping of the edgewise vibrational modes of the blades may be very low or even negative, causing large vibrations and corresponding oscillatory loads. To prevent this, some blades are fitted with vibration dampers. These consist of mass-spring-damper system tuned to vibrate at the resonant frequency. Enter the mass, frequency and damping factor (fraction of critical damping), as well as the damper position and the direction of vibration relative to the local chord line. For the direction, the sign convention is the same as for blade twist[3.2].

3.3.4 Blade icing

The blade mass distribution may be increased to represent the accretion of ice on the blades. The ice mass is calculated according to the specification of Germanischer Lloyd [1] for the machine rotating with all blades iced. Select **Blades iced**, and enter the ice density ρ and the tip chord C_{min}. This should be "linearly extrapolated from the blade contour" according to [1]. For the outer half of the radius, the additional mass per unit length is given by

$$\left(0.00675 + 0.3\exp(-0.32R)\right)\rho c_{min}(c_{min} + c_{max})$$

where R is the rotor radius in metres, and c_{max} is the maximum chord, obtained from the blade geometry[3.2] information.

Between the rotor centre and the half radius point, the additional mass per unit length changes in proprtion to radius, down to zero at the rotor centre.

The change in mass is reflected in the **Turbine Info** screen – click the **Mass Totals** button to see this.

3.4 Blade stiffness distribution

For calculations involving blade flexing or vibrational dynamics[7.1] it is necessary to define the stiffness distribution of the rotor blades. Click the **Stiffness** checkbox **off** if calculations not involving blade flexing are desired and the stiffness data has not been defined.

The stiffness must be defined in both edgewise and flapwise directions at each blade station[3.1]. The stiffness is the product of the Young's Modulus for the material and the second moment of area for the edgewise or flapwise direction as appropriate. The principal axes are assumed to be parallel and perpendicular to the chord line. It is further assumed that they intersect on the pitch axis, and that there are no cross-coupling terms.

To enter or edit data, click on the appropriate entry, or move to it with the arrow keys and press Return.

3.5 Flap-Twist Coupling

Flap-twist coupling allows the twist of the blade to change as a function of the flapwise deflection. It does not introduce any additional torsional degree of freedom, so purely torsional modes are not modelled. Such modes generally have a relatively high frequency, but if there is any purely torsional with a frequency lower than any of the modes being considered, *Bladed* will issue a warning as the resulting behaviour would then be unrepresentative.

For calculations involving blade flap-twist coupling it is necessary to define the **mass moment of inertia per unit length**, the **torsional rigidity**, which is the product of the shear modulus and the torsional constant, and the position of the **principal neutral axis** for bending, as a percentage of chord measured from the leading edge.

To enter or edit data, click on the appropriate entry, or move to it with the arrow keys and press Return.

3.6 Aerofoil sections

At each blade station[3.1] , the aerodynamic characteristics of the blade section must be defined so that lift and drag coefficients may be calculated. Pitch moment coefficients may also be calculated if desired.

Aerodynamic characteristics are defined empirically, by means of look-up tables of lift, drag and pitch moment coefficients tabulated against angle of attack. One such look-up table is referred to here as an aerofoil dataset[3.7]. A number of aerofoil datasets may be combined to define one **aerofoil section**. Two types of aerofoil section may be defined:

Normal Sections:[3.9] for any fixed or pitchable part of blade with no flaps or ailerons. Separate aerofoil datasets may be provided for up to three fixed Reynolds numbers. Interpolation between these datasets will occur depending on the precise Reynolds number at any point in the calculation. For each of the fixed Reynolds numbers, aerofoil datasets may be provided for two different thickness/chord ratios, one either side of the actual thickness/chord ratio for the blade station in question. In this case, a further interpolation will be carried out to give aerodynamic coefficients appropriate to the actual thickness/chord ratio.

Aileron Sections:[3.10] for any parts of the blade which have flaps, ailerons, airbrakes or other aerodynamic control[4.1] surfaces which alter the aerodynamic profile of the blade section. Separate aerofoil datasets may be provided for a number of different deployment angles of the control surface. Interpolation between these datasets will be used to give aerodynamic coefficients appropriate to the actual deployment angle at any point in the calculation.

3.7 Aerofoil datasets

An aerofoil dataset is a look-up table of Lift coefficient, Drag coefficient and Pitching moment coefficient (optionally) as a function of angle of attack. Such tables are usually obtained empirically for particular aerofoil sections at particular Reynolds numbers, for example in wind tunnel tests.

3.7.1 Defining aerofoil datasets

Bladed maintains a database of aerofoil datasets. Click the **Aerofoil** icon on the toolbar for access to the database. As well as the table of aerodynamic coefficients, each aerofoil dataset is characterised by a set of general data giving the characteristics of the aerofoil section to which it applies, namely:

- **Name**: this is an the identifier for referencing the dataset within ***Bladed***.
- **Comments**: to identify the source of the data and the full name of the aerofoil section.
- **Thickness to chord ratio** for the aerofoil section.
- **Reynolds number** for which the dataset applies.
- **Pitching moment centre**: the point about which the aerodynamic pitching moment is defined, as a percentage of the chord backwards from the leading edge.
- **Deployment angle** (for aileron sections *3.10*).
- **Include pitching moment**: set to **No** if the pitching moment data is not available. If any dataset used on the turbine does not have pitching moment data defined, then no pitching moments will be calculated for that turbine.

If possible, give angles of attack ranging from -180° up to 180° as the full range will be needed for some calculations.

3.7.2 Importing a dataset

Aerofoil datasets may be imported from an ASCII file of aerofoil data by clicking the **Import** button (see Section 3.8 for the required format of the ASCII file). A standard file selection screen appears. Select the drive, directory and file where the data is stored. If the identifier specified in the file duplicates one which exists in the database, you will be prompted for a different identifier to be used.

3.7.3 Adding a new dataset manually or via the clipboard

Click the **New** button, fill in the **General data** to characterise the aerofoil dataset, add comments as required, and give the dataset an appropriate identifier in the **Name** box.

The aerofoil data itself may be pasted in from the clipboard by clicking the **Paste** button. The data should be copied from a spreadsheet or tab-delimited file, and should consist of three or four columns: angle of attack in degrees, Lift coefficient, Drag coefficient, and Pitching moment coefficient if included.

Alternatively, type the data for each angle of attack in the coefficients box, clicking **Add** after each one.

Then press **Save**. New datasets may also be created by editing existing ones, as described below.

3.7.4 Editing an existing dataset

Click the **Load** button to list the names of all currently available datasets. Clicking one of these names displays the main characteristics of that dataset, which can then be selected by double-clicking or pressing **OK**. The data for that aerofoil may then be edited. To add a new angle of attack, enter it in the coefficients window and press **Add**. To modify a datapoint, click on it and press **Edit** to make it appear in the coefficients window where it can be edited. To remove a datapoint, click on it and press **Delete**. When editing is finished, click the **Save** button. To save it as a new aerofoil dataset, modify the General data as appropriate, change or insert any comments in the **Comments** box, change the **Name** as required, and then click **Save**.

The **Copy** button may also be used to transfer aerofoil data via the clipboard to another application.

3.7.5 Removing a dataset

Click the **Delete dataset** button.

3.7.6 Viewing aerofoil data graphically

Select the view required, change the range of angles of attack if desired, and click **View Data**. The graph may be printed or saved as a Metafile or copied to the clipboard for subsequent inclusion in a report.

3.8 Format of ASCII aerofoil files

In order to import aerofoil data into the database, generate an ASCII file containing the data, using the format described here. This consists of a header section describing the aerofoil and the conditions to which the data relate, followed by a table of the lift, drag and (optionally) pitch moment coefficients against angle of attack, and terminated by a label which marks the end of the dataset.

Apart from comment line(s), the header lines each contain an identifier, which must be in upper case, followed by an identifier consisting of up to 32 alphanumeric characters:

REFNUM	up to 32 characters	A unique identifier for this dataset
*		Any comment lines here, starting with an asterisk (*), will be incorporated in the database
*		
*		
*		
XA	<real> %	Pitch moment centre, back from leading edge
THICK	<real> %	Thickness/chord ratio
REYN	<real>	Reynolds number
DEPANG	<real> degrees	Aileron deployment angle
NALPHA	<integer>	Number of angles of attack listed in the table
NVALS	<integer> 2 or 3	Number of coefficients: 2 if just lift and drag coefficients are available, 3 if pitch moment coefficients are also present

The table of coefficients follows, consisting of **NALPHA** lines of data each containing:

Angle of attack (degrees), Lift coefficient, Drag coefficient [, moment coefficient if NVALS=3]

with the values on the line separated by commas, spaces or tabs. Finally, there should be a line consisting of the termination string:

ENDSECTION

A very simple example for an ASCII aerofoil database file, just to illustrate the format required, is given here:

```
REFNUM    SET1
* Example dataset
* Use these lines for a description
XA        25.0
THICK     18.0
REYN      3.E6
DEPANG    0.0
NALPHA    12
NVALS     3
-180    0       0       0
-20     -1      0.3     0.037
-12     -0.88   0.03    -0.067
-8      -0.48   0.0107  -0.078
-4      -0.02   0.0073  -0.089
0       0.45    0.0061  -0.104
4       0.891   0.0069  -0.121
8       1.323   0.016   -0.119
12      1.728   0.0218  -0.116
16      2.119   0.07    -0.111
20      2.478   0.225   -0.112
180.0   0.0     0.0     0.0
ENDSECTION
```

3.9 Defining normal aerofoil sections

A normal aerofoil section should be defined for every blade station which is either:

- **Fixed** (see Blade geometry[3.2]) or
- **Moving** (see Blade geometry[3.2]) and pitchable (see Control surfaces[4.1]).

Normal aerofoil sections allow interpolation on up to three Reynolds numbers and up to two thickness/chord ratios - see Aerofoil sections[3.6]. A very flexible interpolation scheme is used, so for example it is not necessary for the individual datasets to have matching angles of attack.

A number of different aerofoil sections may be defined, for use at different blade stations.

On the **Blade properties** window, click on the foil section to be defined. A drop-down list allows an already-defined section to be selected, or select **Define...** to open the **Define Aerofoil Sections** window. This allows the characteristics of already-defined sections to be viewed or edited, or new ones created by clicking **New**.

To view or edit an existing aerofoil section:
In the **Define Aerofoil Sections** window, select the section number required.

To set up a new aerofoil section:
In the **Define Aerofoil Sections** window, press **New** to start a new foil section.

To edit an aerofoil section:
Up to six aerofoil Dataset [3.7] names may be entered in the boxes provided. Click on any white or red box to open the **Aerofoil Dataset Selection** window. This presents a list of all suitable datasets from the database (they must be in ascending order of Reynolds number and thickness). If desired, the selection criteria may be modified to further restrict the list displayed. Select a dataset from the list and click **OK**.

If fewer than six boxes are required, the boxes used must form a rectangular pattern so that the interpolation scheme is fully defined. The top left hand box must always be used.

To remove an aerofoil section:
There is no need to delete an aerofoil section if it is no longer needed at any blade station. The sections which are still in use will be renumbered when the **Blade Properties** window is closed, starting from 1. The aerofoil datasets themselves of course remain in the database.

3.10 Defining aileron sections

An aileron section should be defined for every blade station which is defined as **Moving** (see Blade geometry[3.2]) if **ailerons** are specified for the aerodynamic control surfaces[4.1]. Flaps and airbrakes can be treated as if they are ailerons.

Aileron sections allow interpolation between a set of aerofoil datasets[3.7] defined for different aileron deployment angles.

A number of different aileron sections may be defined if required, for use at different blade stations.

On the **Blade properties** window, click on the foil section to be defined. A drop-down list allows an already-defined section to be selected, or select **Define...** to open the **Define Aerofoil Sections** window. This allows the characteristics of already-defined sections to be viewed or edited, or new ones created by clicking **New**.

To view or edit an existing aerofoil section:
In the **Define Aerofoil Sections** window, select the section number required.

To set up a new aerofoil section:
In the **Define Aerofoil Sections** window, press **New** to start a new foil section.

To edit an aerofoil section:
Click **Add** or **Insert** to increase the number of aerofoil datasets[3.7] to interpolate between. Click **Delete** to remove one. Double-click on any entry to open the **Aerofoil Dataset Selection** window. This presents a list of all suitable datasets from the database (they must be in ascending order of deployment angle). If desired, the selection criteria may be modified to further restrict the list displayed. Select a dataset from the list and click **OK**.

No entries may be left blank.

To remove an aerofoil section:
There is no need to delete an aerofoil section if it is no longer needed at any blade station. The sections which are still in use will be renumbered when the **Blade Properties** window is closed, starting from 1. The aerofoil datasets themselves of course remain in the database.

4. DEFINING THE REST OF THE TURBINE

Having defined the turbine blades, this section describes how to build up a model of the complete turbine structure including the rotor and hub, power train, tower and nacelle.

4.1 Defining the rotor

Click the **Rotor** icon on the toolbar to define the basic characteristics of the turbine rotor and hub. The following data must be defined:

- **Rotor diameter**
- **Number of blades**
- **Hub height**: from the ground to the centre of the rotor (i.e. the intersection of the blade and shaft axes).
- **Tower height**: from the ground to the top of the tower. See also: Tower[4.4].
- **Blade set angle**: the angle at which the blade is mounted onto the hub, i.e. the angle between the blade reference plane[3.2.1] and the plane (or cone) of rotation of the rotor. More positive values of set angle push the leading edge further upwind. It is usually convenient to define the set angle as zero, but particularly for stall regulated machines the set angle provides a simple way of rotating the whole blade without having to re-define the twist distribution.
- **Cone angle**: the angle between the blade axis and the rotor plane (normally positive).
- **Tilt angle**: the angle between the shaft and the horizontal (normally positive).
- **Overhang**: the horizontal distance between the rotor centre and the tower centreline.
- **Lateral offset**: the horizontal offset between the shaft and tower axes.
- **Rotational sense**: the turbine may rotate clockwise or anti-clockwise when viewed from upwind.
- **Position**: the rotor may be upwind or downwind of the tower in normal operation.
- **Speed type**: the turbine may be defined as a fixed or a variable speed turbine. See also Generator[4.8] , Control[5].
- **Control surfaces**: specifies whether the blade is fixed, pitchable (includes full or partial span pitch) or aileron controlled (includes any type of flap or airbrake). See also Blade Geometry[3.2], Control[5], Start-up[5.12], Normal stop[5.13], Emergency stop[5.14].
- **Transmission**: select Gearbox or Direct Drive. For a direct drive system, either set the gearbox ratio[4.6] to 1, or select the Direct Drive option here. If the Direct Drive option is selected, the generator mass is separately identified on the Hub screen[4.2], otherwise it must be included in the Nacelle mass[4.11].
- **Cut-in wind speed**: the steady wind speed at which the turbine is brought on or off line in low winds.
- **Cut-out wind speed**: the steady wind speed at which the turbine is brought on or off line in high winds.

Clicking the **Hub** tab allows the hub configuration to be defined. This is described in Section 4.2.

4.2 Defining the hub

From the **Rotor**[4.1] screen, click the **Hub** tab to define the hub characteristics.

Enter the **Spinner diameter.** This is the diameter of any spinner or nose-cone, within which the blades themselves experience no aerodynamic forces.

Blade root section
The blade root section connects the blade root to the shaft. It is assumed to be cylindrical in section. It is defined by:

- **Length**: this defines the actual radius of the first blade station[3]. Set this to zero if there is no root section. The rotor diameter[4.1] must be twice the sum of the root section length and the blade length, defined by the radial position of the last blade station.
- **Diameter**: the diameter of the cylindrical tube which forms the blade root section.
- **Drag Coefficient**: the drag coefficient to be used for any part of the blade root section which is outside the spinner. Zero lift coefficient is assumed.

If the spinner entirely covers the root section, the diameter and drag coefficient values will not be used.

For one and two bladed rotors, click the check box to specify a teetered hub if required. See also Options[7.2.3]. For teetered hubs, enter a **Delta-3 angle** if pitch-teeter coupling is required. A positive delta-3 angle acts to stabilise teeter by increasing the angle of attack when the blade teeters into the wind. A **Special** setting is available in case other forms of pitch-teeter coupling are required, by special arrangement with Garrad Hassan.

Teeter restraint
If a teetered hub is selected, a teeter restraint system may be specified by clicking the Teeter restraint[4.3] button.

Mass information
For dynamic calculations, check the **Mass** checkbox to allow the necessary masses and inertias to be specified as follows:
- **Hub mass**: the mass of the hub, including the spinner and any blade root section.
- **Hub mass centre**: the distance from the intersection of the shaft and blade axes to the centre of mass of the hub, in a direction measured away from the tower.
- **Hub inertia**: the moment of inertia of the hub mass about the shaft axis.

For direct drive systems, enter also the generator mass information as follows:
- **Generator mass**
- **Generator mass centre**: the distance between the centre of mass of the generator and the centre of mass of the hub.
- **Generator inertia**: the moment of inertia of the generator. Note: this is **always required**, although not actually used for static calculations.

and for one-bladed rotors only:
- **C/Weight Mass**: the mass of the counterweight.
- **C/Weight Inertia**: the moment of inertia of the counterweight about the shaft axis.

Note

The **Turbine Information** window shows the total masses and inertias of all turbine components whose mass characteristics are defined. Click **Mass Totals,** or use the **Windows** pull-down menu on the main toolbar to open this window.

4.3 Teeter restraint

A spring and damper model of a teeter restraint system is provided. If a teetered rotor has been specified, click the **Teeter restraint** button on the **Hub** screen, select **Standard model**, and enter the following parameters:

- **Free teeter angle**: The teeter restraint only start to act when this teeter angle is exceeded.
- **Spring preload**: The torque required to start moving the teeter restraint.
- **Spring stiffness**: The rotational stiffness of the teeter restraint.
- **Teeter damping**: The rotational damping of the teeter restraint.

Click **Special** if a client-specific teeter restraint model has been provided by arrangement with Garrad Hassan, and enter the appropriate parameters.

See also: Calculation options[7.2.3].

4.4 Defining the tower

Click the **Tower** button on the toolbar to define the tower characteristics. The tower diameter is specified as a function of height. The mass and stiffness distributions are also required if tower vibration is to be modelled. It is also possible to specify the mass and stiffness properties of the foundation[4.4.4], as this can influence the tower vibrational modes significantly.

4.4.1 Tower

The tower is modelled by defining its characteristics at a number of stations from the tower base to the tower top. Tower properties must be defined for at least two tower stations (the tower base and tower top), although if tower vibrations[7.1] are to be modelled, a minimum of 5 stations is recommended in order to achieve a reasonable degree of accuracy.

Check the **Tower geometry** check-box to define the tower dimensions. These are used to calculate the tower shadow[6.3] and windage loads.

Click the **Add** button to add a new tower station. Stations are automatically sorted by height. To highlight a station, click on the station number or a graph point. Click **Delete** to remove a highlighted station.

To enter or edit data, highlight the data item by clicking on it, or by moving to it using the arrow keys and then pressing the Return key. Press Escape to restore the previous value.

Highlight a block of cells by dragging the mouse. These cells may be copied to the clipboard, or the contents of the clipboard may be pasted in. In this way, data from a spreadsheet or a tab-delimited ASCII file may be directly pasted in. Click **Undo** to reverse a paste operation.

Enter the tower station height and tower diameter for each station. In the case of an onshore[4.4.3] turbine the station heights are defined relative to ground level, whilst for an offshore[4.4.3] turbine, station heights are defined relative to the mean water level.
If the first tower station is above the ground or sea bed level, the tower is assumed to be mounted on a rigid pedestal, as indicated on the **Tower Geometry** diagram. If the height of the first tower station is below the ground or sea bed level, the tower foundation is assumed to be buried as indicated, and no external forces are assumed to act on the buried portion of the tower.

For all cases, the height of the top tower station must correspond to the tower height defined on the Rotor[4.1] screen.

The **Show** button controls the graphical display of tower properties. The graph is updated as soon as any new values are assigned, giving an instant indication if faulty values are entered. Controls are provided to print the graph or save it to a Metafile, which can subsequently be incorporated into a report.

Check the **Mass** check-box to allow the mass per unit length to be entered at each tower station. This is necessary for correct calculation of the tower base loads, and also for modelling tower vibrations[7.1] , in which case it is also necessary to check the **Stiffness** check-box and supply bending stiffness values at each station. These are defined as the product of the Young's modulus and the second moment of area.

Alternatively, for a tower of circular cross-section, the mass and stiffness distributions can be calculated automatically by entering the wall thickness at each station. Type the name of a material and enter its density and Young's modulus in the boxes provided. A number of different materials may be entered if desired. Then select the appropriate material for each station.

If a discontinuity is to be defined, for example a step change in wall thickness or other property, simply enter two stations at exactly the same height to define the discontinuity. Do not enter very closely spaced stations.

4.4.2 Flanges and point masses

Flanges should be modelled as point masses, not by an increase in wall thickness between closely-spaced stations. Click the **Point masses** button to define the height and additional mass of each flange or other point mass.

4.4.3 Environment

Click either **Land** or **Sea** in the **Environment** panel to define whether the turbine is sited onshore or offshore. The **Tower Geometry** diagram will then show either the ground level or the sea bed and mean water level, as appropriate. To change the water depth relative to the base of the tower, change the **Depth of first tower station** as appropriate.

Enter the aerodynamic drag coefficient to enable windage forces and aerodynamic damping to be calculated. For an offshore turbine, enter also the hydrodynamic drag and inertia coefficients, to enable wave and current forces and hydrodynamic drag to be calculated, and the mean sea depth. For a description of wave and current loading, see the Theory Manual.

4.4.4 Foundation

If the tower mass and stiffness check-boxes have been checked, it is also possible to define the mass and stiffness properties of the foundation, which can have a significant influence on the tower vibrational properties.

Use the check boxes to specify whether translational and/or rotational motion at the tower base are to be taken into account. If translation is selected, enter the foundation mass and translational stiffness in the appropriate boxes. If rotation is selected, enter the foundation moment of inertia and rotational stiffness. If neither is selected, a rigid foundation is assumed.

Note
The **Turbine Information** window shows the total masses and inertias of all turbine components whose mass characteristics are defined. Click the **Mass totals** button or use the **Windows** pull-down menu on the main toolbar to open this window.

4.5 Defining the power train

Click the **Power Train** button on the toolbar to define the following aspects of the power train:

- the transmission[4.6] (gearbox and shafts),
- any flexible mountings[4.7],
- the generator[4.8],
- the energy losses[4.9], and
- the network[4.10] to which the turbine is connected.

4.6 Transmission

The **Power Train** button on the toolbar allows the drive train or transmission to be specified.
The transmission consists of the gearbox and shafts. Two alternative ways of specifying the drive train are provided:

- Locked speed model[4.6.1]
- Dynamic model[4.6.2]

4.6.1 Locked speed model

This model is available for simple calculations in which the turbine rotor is assumed to rotate at an absolutely constant speed. The rotor speed is specified along with the gearbox ratio, but no other characteristics of the drive train nor of the generator may be specified. Flexible mountings cannot be modelled either. This model is useful for initial aerodynamic design of fixed speed rotors, but for detailed performance and loading calculations the dynamic model is more appropriate.

4.6.2 Dynamic model

This model includes the rotational degree of freedom of the rotor. Torsional flexibility of the drive train may be included as an option.

The gearbox ratio must be specified, along with the moment of inertia of the generator rotor including the high speed shaft and any brake disc etc. Note that the moment of inertia of the turbine rotor is defined by the blade mass distribution[3.3] and the hub inertia[4.2].

The rotor speed will be governed by the characteristics of the generator[4.8], which must be defined in this case. For a variable speed turbine it is also necessary to define an appropriate controller[5] to define the speed schedule. If there is a shaft brake[5.11] to be used in stopping and parked calculations, specify where it is located on the high speed or low speed shaft. Click the **Brakes** button to specify the dynamic characteristics of the brakes.

Use the check-boxes to specify if there is significant torsional flexibility in the low speed and/or high speed shaft, and supply the torsional stiffness. Torsional damping may also be specified. For fixed speed turbines, the damping is unlikely to have much effect as it will be swamped by the damping provided by the induction generator. In fact the damping may be sufficient to ensure that there is very little drive train torsional motion, in which case it may be advantageous not to specify any shaft torsion, as the simulations will then run faster.

The dynamic drive train model may be used in combination with flexible mountings[4.7].

Note: including drive train flexibility may reduce the speed of simulations.

4.7 Drive train mountings

The **Power Train** button on the toolbar also allows the drive train mountings to be specified.

If desired, torsional flexibility may be specified either in the gearbox mounting or between the pallet or bedplate and the tower top. This option is only allowed if the dynamic drive train model[4.6.2] is specified.

In either case, specify the following parameters:

- The torsional stiffness of the mounting.
- The damping of the mounting.
- The moment of inertia of the moving components about the low speed shaft axis. In the case of a flexible gearbox mounting, this is the moment of inertia of the gearbox casing. In the case of a flexible pallet mounting, it is the moment of inertia of the gearbox casing, the generator stator, the moving pallet and any other components rigidly fixed to it.

If either form of mounting is specified, the direction of rotation of the generator shaft will affect some of the loads. If the low speed and high speed shafts rotate in opposite directions, specify a negative gearbox ratio in the drive train[4.6] model.

Flexible mountings introduce a high frequency mode which is liable to result in much slower simulations. If the mountings are relatively stiff, for example if they are intended for noise isolation, then they will not influence loads significantly and are best omitted. If they are intended to modify the rotational dynamics, then they can be approximated by a corresponding reduction in shaft stiffness. This will have little effect on rotor and tower loads, but simulations will run faster.

4.8 Generator

The **Power Train** button on the toolbar also allows the generator to be specified. The generator characteristics must be modelled if the dynamic drive train model[4.6.2] is specified.

Three generator models are provided:

- Induction generator[4.8.1] (for constant speed and two-speed turbines)
- Variable speed generator[4.8.2] (for variable speed turbines)
- Variable slip generator[4.8.3] (for variable slip turbines)

See also Defining the rotor[4.1], Control[5]).

4.8.1 Induction generator

This model represents an induction generator directly connected to the grid. It is characterised by:

- **Rated slip**: the slip speed at rated power as a percentage of synchronous speed.
- **Rated power**: the power output corresponding to rated slip.
- **Synchronous speed**: the rotational speed of the generator rotor at zero load.
- **Generator time constant**: this is the short-circuit transient time constant for the generator. **Note**: a small time constant may result in slower simulations. If it is very small, specifying a zero time constant will speed up the simulations, without much effect on accuracy.

The following table shows the synchronous speed for some typical cases:

Grid frequency:	50 Hz	60 Hz
2 pole generator:	3000 rpm	3600 rpm
4 pole generator:	1500 rpm	1800 rpm
6 pole generator:	1000 rpm	1200 rpm
8 pole generator:	750 rpm	900 rpm

Alternatively, click **Electrical dynamics** for a detailed representation which models the generator starting from its equivalent circuit parameters, namely stator and rotor resistance and inductance, mutual inductance, number of poles, grid frequency and voltage. An auxiliary load may be specified in parallel with the generator to represent any power-consuming auxiliary equipment which the turbine may have. Power factor correction capacitors may also be modelled. This representation of the generator allows both active and reactive power to be calculated, as well as voltages and currents.

For starting the generator, click **Soft start etc**… to specify a ramp time which a soft starter (if there is one) will use to ramp up the voltage from zero to the working value. It is also possible to specify that the power factor correction capacitors are switched in in stages, with a delay between each stage.

A choice of generator model orders is provided. Choose a high order for higher accuracy, or a low order for faster simulations. Further details are provided in the Theory Manual.

Since this representation includes modelling of the electrical losses in the generator and auxiliaries, electrical losses may not also be specified under Energy losses[4.9].

The test box at the bottom may be used to enter a slip value, allowing the power and power factor to be checked on both sides of the auxiliary load and power factor correction capacitors.

If **Two-speed operation** has been selected, enter all the relevant parameters for both the main generator and the low speed generator. The code automatically determines which generator to use, as follows:

- Steady calculations and simulation initial conditions: if the wind speed is less than 13 m/s, the generator which gives the higher power is used, otherwise the main (high speed) generator is used.
- Start-ups using built-in start logic: the generator whose synchronous speed is closer to the speed specified for generator switch-on[5.12] is used.

An external controller[5.8] can be used to determine which generator to use, and to implement speed change logic.

4.8.2 Variable speed generator

This model should be used for a variable speed turbine. It models both the generator and frequency converter. A variable speed turbine requires a controller[5] which generates a torque demand. The variable speed generator is modelled as a first order lag from demanded torque to actual generator torque, independently of rotational speed. This is specified by the **Power electronics time constant**. **Note**: a small time constant may result in slower simulations. If it is very small, specifying a zero time constant will speed up the simulations, without much effect on accuracy.
The **minimum and maximum generator torque limits** must be specified. Motoring may occur if a negative minimum torque is specified. The **phase angle** defines the phase relationship between the current and voltage from the network side of the frequency converter, and therefore the power factor. It is assumed that the power factor is completely controllable by the frequency converter.
Drive train damping feedback: the generator torque control may also include a built-in feedback term derived from the generator speed by means of a transfer function. This is superimposed on the torque demand from the controller, and may be used to help damp out drive train torsional vibrations. Specify the transfer function required in terms of numerator and denominator polynomials in the Laplace operator.

4.8.3 Variable slip generator

A variable slip generator is an induction generator with a controlled resistance in series with the generator rotor windings, giving a fixed speed induction generator characteristic below rated and limited range variable speed operation above rated. The parameters required are a combination of the fixed speed induction generator parameters and the variable speed generator parameters detailed above, although power factor control is not available. The maximum slip should also be supplied. Alternatively the electrical dynamics model can be selected, in which case the following additional parameters are required to define the variable slip control:

Additional controlled resistance: the additional resistance which is connected in series with the generator rotor, and controlled to vary the slip,
Proportional gain: the proportional gain of the PI controller which adjusts the additional resistance in response to the normalised rotor current error, and
Integral gain: the integral gain of the PI controller.

If a two-speed variable slip generator is selected, torque control is only possible for the main generator.

A variable speed pitch regulated controller[5.4] should be selected for use with variable slip generators - see also Variable speed control below rated[5.7].

4.9 Energy losses

The **Power Train** button on the toolbar also allows the energy losses to be specified. Power train energy losses are divided into mechanical and electrical losses.

4.9.1 Mechanical losses

A choice of two options is available to specify mechanical losses, for convenience. Select **Expressed as torque** to specify a loss torque in kNm, or **Expressed as power** to specify a power loss in kW. Then enter either a constant loss in the box, or click the appropriate **Add** button(s) as required to create a look-up table. This may be a simple look-up table on shaft speed or on shaft torque or power (depending on the option selected), or a two-dimensional look-up table on shaft speed and either torque or power. If a look-up table is used, loss values will be linearly interpolated from the table.

The option to express losses in terms of power has been provided for convenience, since data is sometimes provided in this form. However this model is inappropriate at low rotational speeds. If any calculations at low speed are to be carried out, including start-ups, shut-downs, idling, or parked simulations with slipping brake, then it is advisable to recalculate the losses in terms of torque.

4.9.2 Electrical losses

Two alternative models are available for the electrical losses:

Linear model:

This requires a no-load loss L_0 and an efficiency ε, where the electrical power output P_e is related to the generator shaft input power P_s by:

$$P_e = \varepsilon (P_s - L_0)$$

Look-up table:
The power loss $L(P_s)$ is specified as a function of generator shaft input power (P_s) by means of a look-up table. The electrical power output (P_e) is given by:

$$P_e = P_s - L(P_s)$$

Use the **Add** button to add points to the look-up table. Double-click on a look-up table entry to edit it.

4.10 The Electrical Network

From the **Power Train** button on the toolbar, click **Network** to define the electrical characteristics of the windfarm network and the connection from the windfarm to the external network. This is useful if the **Electrical dynamics** option has been selected for the fixed speed induction generator, or if the phase angle is specified for the variable speed generator. Then the network currents and voltages, as well as active and reactive power, will be calculated at the turbine, the point of common coupling, and the 'infinite busbar'.

Click **Network as shown**, and enter the resistance and inductance of the line connecting the turbine to the point of common coupling, and of the line connecting the windfarm to the infinite busbar. Enter also the voltage for which these impedance values were calculated. If required, specify the number of additional turbines connected at the point of common coupling. These turbines will be assumed identical to the turbine being modelled, and running at a constant current and voltage equal to that of the turbine being modelled at the start of the simulation. The dynamics of the additional turbines is not modelled, but the steady state voltage rise at the point of common coupling will be calculated.

4.11 The nacelle

Click the **Nacelle** icon on the toolbar to define the nacelle characteristics.

Check the **Geometry** checkbox to define the characteristics required to calculate windage loads. Two models are available:

Drag model: the following information is required:
- **Nacelle length**
- **Nacelle height**
- **Nacelle width**
- **Nacelle drag coefficient**.

And the drag force is then calculated based on the projected nacelle area perpendicular to the wind direction at any time,

Aerofoil model: if the lift and drag forces on the nacelle are known as a function of yaw angle, specify:
- **Nacelle length**
- **Nacelle height**
- **The position of the upwind end of the nacelle relative to the tower axis**, and
- Select an **aerofoil from the database** which has been constructed to give the correct variation of lift and drag with angle of attack (yaw angle). The nacelle length will be used as the aerofoil chord, and the aerofoil area will be the product of length and height.

Check the **Mass** checkbox to define the nacelle mass characteristics required for dynamic calculations. This includes the nacelle structure and all the machinery within it. It does not include the rotor blades and hub. If the Direct Drive[4.1] option is selected, the mass of the generator[4.2] is also excluded. Enter the following:

- **Mass on tower top**.
- **Centre of mass position** - see diagram.
- **Moment of inertia about tower axis**.

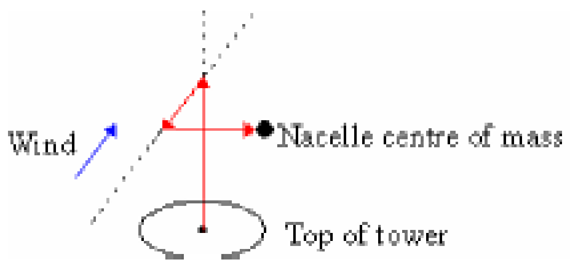


Diagram showing positive directions for centre of mass position relative to the top of the tower, showing wind direction during normal running.

Note
The **Turbine Information** window shows the total masses and inertias of all turbine components whose mass characteristics are defined. Use the **Windows** pull-down menu on the main toolbar to open this window.

5. CONTROL SYSTEMS

Click the **Control** icon on the toolbar to open the **Control Systems** screen. This allows definition of the power production control characteristics as well as supervisory control functions.

Power production control

Power production control models are available for pitch or stall regulated, fixed or variable speed turbines.

Select one of the following options:

1. **Fixed speed stall regulated**[5.1] : the induction generator model should be selected.
2. **Fixed speed pitch regulated**[5.2] : the induction generator model and a pitch or aileron controlled rotor should be selected.
3. **Variable speed stall regulated**[5.3] : the variable speed generator model should be selected.
4. **Variable speed pitch regulated**[5.4] : the variable speed or variable slip generator model should be selected, and a pitch or aileron controlled rotor.

See: Generator[4.8], Drive train[4.6] , Defining the rotor[4.1].

The steady state parameters define the operating envelope of the turbine, and are required for calculation of the steady power curve and steady operational loads. Click on **Controller Dynamics** to define additional parameters which model the dynamic action of the controller, such as controller gains and transducer and actuator time constants.

The different power production control models are described in Sections 5.2 to 5.7 below. Further details of the power production controllers is provided in the Theory Manual. It is also possible for user-defined control logic to be used - see Section 5.8.

Supervisory control

The Supervisory control screens may also be accessed by clicking the **Control** icon on the toolbar.

Use the Supervisory control screens to define the characteristics of the following:

- Shaft brake[5.11]
- Start-up sequence[5.12]
- Normal stop sequence[5.13]
- Emergency stop sequence[5.14]
- Idling conditions[5.15]
- Parked conditions[5.16]
- Yaw control.[5.17]
- User-defined controllers. [5.8]
- Pitch actuators. [5.10]

In-built supervisory control logic is available for stops and starts, but the user may also supply the required logic for these[5.8], as well as for yaw control.

5.1 Fixed Speed Stall Regulated Control

There is no in-built controller for fixed speed stall regulated turbines, so no parameters need be entered in this case.

However, clicking on **Dynamics** will allow the speed and power transducer dynamics to be defined, even though these are not used for power production control. The transducer signals are available so that an external controller can have access to them if desired. Click on **Use external controller for pitch/airbrake control** if an external controller will be used to control any pitchable blades, tips or ailerons. In this case, the aerodynamic control surfaces must be defined appropriately on the **Rotor** and **Blade** screens. The software rate pitch limits will be available to the external controller if required, and the pitch actuator dynamics should be specified.

5.2 The fixed speed pitch regulated controller

Use this control model for a fixed speed turbine (i.e. with a directly-connected induction generator[4.8.1]) which has a pitch- or aileron-controlled rotor[4.1].

5.2.1 Steady state parameters

To specify the steady state operating curve, supply the following data:

- **Demanded electrical power**: the power set-point, i.e. the electrical power output required above rated wind speed.
- **Minimum pitch angle**: the lower limit of pitch or aileron deployment angle.
- **Maximum pitch angle**: the upper limit of pitch or aileron deployment angle.
- Select **Pitch feathering** or **Assisted stall**: to specify the direction of pitching. If Pitch Feathering is selected, the Minimum pitch angle will be set for normal operation below rated wind speed, and the pitch (or aileron) angle will increase above rated. If Assisted Stall is selected, the Maximum pitch setting is used below rated, and the pitch (or aileron) angle decreases (or moves to more negative values) above rated.

5.2.2 Dynamic parameters

This controller varies the pitch or aileron angle above rated, to regulate the electrical power output to the set point given above. The following information is needed for simulations which include controller dynamics:

- **Power transducer time constant** — See transducers[5.9]. A speed transducer may also be defined if desired.
- **Pitch actuator dynamics** — Enter the software rate limits for use in the control algorithm. Click **Actuator...** to specify the pitch actuator[5.10] details.
- **Control algorithm** — Choose between PI[5.5] and Discrete External[5.8], and click **Define...** to specify the details.

5.3 The variable speed stall regulated controller

Use this control model for a variable speed turbine (i.e. with a variable speed generator[4.8]) which uses the aerodynamic stall characteristics of the rotor[4.1] to control the power above rated wind speed.

5.3.1 Steady state parameters

To specify the steady state operating curve, supply the following data:

- **Torque-speed curve below rated** See Variable speed control below rated[5.7].
- **Demanded shaft power** The power set-point, i.e. the shaft power required above rated wind speed. See Energy losses (Section 4.9).
- **Maximum generator speed** Usually the same as the Optimal mode maximum speed, but exceptionally a higher value could be used for high wind speed operation.

5.3.2 Dynamic parameters

This controller varies the generator torque to regulate the speed of the generator and rotor. Above rated, the power can be controlled by adjusting the speed set-point to stall the rotor. The following information is needed for simulations which include controller dynamics:

- **Speed transducer time constant** See transducers[5.9]. A power transducer may also be defined if desired. The in-built control algorithm uses the product of measured speed and generator torque to give a measured shaft power signal.

- **Control algorithm** Choose between PI[5.5] and Discrete External[5.8], and click **Define...** to specify the details. If you choose PI, you must define both the torque and the power controller details. See also Variable speed control below rated[5.7].

As for the fixed speed stall regulated controller[5.1], it is possible to define an external pitch controller and a pitch actuator if required.

5.4 The variable speed pitch regulated controller

Use this control model for a variable speed turbine (i.e. with a variable speed generator[4.8]) which has a pitch- or aileron-controlled rotor[4.1] .

5.4.1 Steady state parameters

To specify the steady state operating curve, supply the following data:

- **Torque-speed curve below rated:** See Variable speed control below rated[5.7].
- **Pitch feathering** or **Assisted stall**: Specify the direction of pitching.
- **Minimum pitch angle**: the lower limit of pitch or aileron deployment angle.
- **Maximum pitch angle**: the upper limit of pitch or aileron deployment angle. If Pitch Feathering is selected, the Minimum pitch angle will be set for normal operation below rated wind speed, and the pitch (or aileron) angle will increase above rated. If Assisted Stall is selected, the Maximum pitch setting is used below rated, and the pitch (or aileron) angle decreases (or moves to more negative values) above rated.
- **Demanded generator torque:** the maximum generator torque demand, used as a constant torque demand above rated wind speed.
- **Demanded speed above rated:** the generator speed demand above rated. This is usually the same as or slightly above the Optimal mode maximum speed.
- **Maximum generator speed:** must be greater than the Demanded speed above rated.

5.4.2 Dynamic parameters

This controller varies the generator torque to regulate the speed of the generator and rotor. Above rated, the torque is held constant and the speed is then controlled by adjusting the blade pitch angle. The following information is needed for simulations which include controller dynamics:

- **Generator speed transducer**: See transducers[5.9]. A power transducer may also be defined if desired.
- **Pitch actuator dynamics**: Enter the software rate limits for use in the control algorithm. Click **Actuator...** to specify the pitch actuator[5.10] details.
- **Torque control algorithm**: Choose between PI[5.5] and Discrete External[5.8], and click **Define...** to specify the details. See also Variable speed control below rated.
- **Pitch control algorithm**: Choose between PI[5.5] and Discrete External[5.8], and click **Define...** to specify the details.

5.5 PI control

PI or Proportional plus Integral controllers are widely used for closed loop controllers on wind turbines. A PI controller with input x and output y has the form:

$y = \mathbf{K_p}.x + \mathbf{K_i}.\text{int}(x)$

where
$\mathbf{K_p}$ = Proportional gain
$\mathbf{K_i}$ = Integral gain
and int(x) is the time integral of x.

The input x is the deviation of a measured quantity from its desired level or set-point. The output y is the demanded control action, which is intended to minimise those deviations.

Bladed has the following built-in PI controllers:

Fixed speed pitch regulated	x = deviation of measured power from the set-point y = pitch angle demand
Variable speed torque control (both stall and pitch regulated)	x = deviation of measured generator speed from the set-point y = generator torque demand
Variable speed stall regulated power control	x = deviation of shaft power from the set-point y = generator speed set-point
Variable speed pitch controller	x = deviation of measured generator speed from the set-point y = pitch angle demand

Various control design techniques are available to help with the selection of the gains $\mathbf{K_p}$ and $\mathbf{K_i}$. However the design of controllers is a specialist task. The main aim is usually to minimise deviations from the set-point without excessive control action and without causing any instabilities. A non-zero integral gain is important to ensure that the steady-state error is zero. This means that the measured quantity settles at the desired value in steady state conditions. The ratio $\mathbf{K_p}/\mathbf{K_i}$ is known as the integral time constant.

It is often useful or necessary to vary the overall gain of the controller depending on the operating point. This is known as Gain scheduling - see Section 5.6.

Garrad Hassan will be pleased to undertake the design of any wind turbine controllers.

The **desaturation time constant** is used to prevent "integrator wind-up" when the output y is constrained by limits. If this is too long, the controller may take too long to break away from a limit. For example a pitch controlled machine crossing rated wind speed may take too long before the pitch starts acting. On the other hand a very short time constant may result in slower simulations. If the PI controller being modelled is actually implemented in discrete form, as is usual, then the desaturation time constant should be chosen to be somewhat smaller than the discrete controller timestep. Alternatively, specify a zero time constant for instantaneous desaturation.

5.6 Gain scheduling

The gains of a closed loop controller such as a PI controller[5.5] are designed to give good control (i.e. stable operation with good tracking of the set-point) at one particular operating point, for example at one wind speed. At a different operating point, the gains may need to be modified because the relevant characteristics of the system may be different. This is particularly true of pitch controlled turbines, where the aerodynamic gain changes significantly with pitch angle, and hence with wind speed.

The gain scheduling facility allows the overall gain of each PI controller to be modified as a function f(V) of some chosen variable V. Thus the actual proportional and integral gains at any point are given by $\mathbf{K_p}$/f(V) and $\mathbf{K_i}$/f(V), where $\mathbf{K_p}$ and $\mathbf{K_i}$ are the proportional and integral gains as entered in the boxes provided.

The variable V is selected by clicking the arrow by **Schedule on:** and choosing from the list. The variables to choose from are:

- **Pitch** [in radians] (for pitch regulated controllers only).
- **Power** [in W]
- **Speed** [in rad/s] (generator speed, for variable speed controllers only)
- **Wind speed** [in m/s] (note that in practice, a suitable wind speed signal is unlikely to be available).

There are three possibilities for f(V):

- **Constant**: the gains are fixed, with f(V) constant as defined by **Value.**
- **Look-up table**: f(V) is defined by a look-up table. Click **LUT Data** to open the table. Use **Add** to add points to the table, and double-click on an entry to edit or enter a value. The X values refer to the variable V in the units shown, and the Y values are the corresponding f(V).
- **Polynomial**: f(V) is defined by a polynomial. Click **Coefficients** to define it. Use **Add** to add terms to the polynomial, and double-click on an entry to edit or enter a value. Enter a coefficient for each term of the polynomial. f(V) is the sum of terms of V (in the units defined above) raised to the power of the order n, and multiplied by the coefficient entered for that term.

Recommendations
For pitch regulated turbines, it is usually useful to apply gain scheduling to the above-rated pitch controller. This applies to both fixed speed and variable speed pitch regulated controllers. This is because the sensitivity of aerodynamic torque to changes in pitch angle is much greater at large pitch angles, and hence large wind speeds, than close to rated. The gain schedule may be defined as a function of pitch angle, and it is often appropriate, though conservative, to set f(*pitch angle*) to be proportional to the partial derivative of aerodynamic torque with respect to pitch angle. Once the steady state control parameters have been defined, use the steady operating loads calculation to calculate the partial derivatives at each operating point, using the '**Calculate pitch and speed change**' option.

Tip: in practice, for full-span pitch-regulated rotors pitching towards feather, setting f(*pitch angle*) to be proportional to pitch angle is often a reasonable first approximation, but this is unlikely to apply to partial span pitch or aileron-controlled rotors, or in the assisted stall case.

Gain scheduling is unlikely to be required for below rated controllers on variable speed turbines. For above-rated stall-regulated variable speed controllers, the need for gain scheduling will depend on the stall characteristics.

5.7 Variable speed control below rated

Select **Optimal tip speed ratio** or **Look-up table**, and proceed as follows:

5.7.1 Optimal tip speed ratio

Below rated wind speed, a variable speed turbine may try to stay at its optimum tip speed ratio wherever possible, by changing the rotor speed in proportion to the wind speed. This maximises the power coefficient and hence the aerodynamic power available.

This can be achieved in the steady state by setting the generator torque to be proportional to the square of the rotor or generator speed. The **Optimal mode gain** multiplies the square of the generator speed to give the required generator torque demand. It can be calculated as:

$$K_{opt} = \pi\rho R^5 C_p / 2\lambda^3 G^3$$

where

K_{opt} = Optimal mode gain

ρ = air density

R = rotor radius

C_p = power coefficient at λ

λ = desired tip speed ratio

G = gearbox ratio

The torque demand is then given by

$$Q_d = K_{opt}\Omega^2$$

where

Q_d = generator torque demand

Ω = measured generator speed

Note that energy output may not be maximised by maximising aerodynamic efficiency, because the energy losses may also vary with the operating point. It may therefore be better to track a slightly different tip speed ratio. The desired tip speed ratio and the corresponding power coefficient should be used in calculating the **Optimal mode gain** as above.

5.7.2 Look-up table

Click **Data**, and enter a look-up table of generator torque demand against generator speed, using the **Add** button to add entries to the table. The **View** button allows the curve to be visualised.

5.7.3 Other parameters

In addition to the above, specify the **Minimum generator speed** and the **Optimal mode maximum speed** (this must be no greater than the **Maximum generator speed**).

The controller will adjust torque demand as above using the optimal mode gain or the look-up table provided the generator speed lies between the **Minimum generator speed** and the **Optimal mode maximum speed**. If either of these speed limits is reached, it is used as the speed set-point for the Speed Control by Torque Demand PI controller. The deviation of measured generator speed from this set-point is the input to the controller, and generator torque demand is the output. Thus the generator torque is varied to keep the generator speed at the limit.

Note: This PI controller remains active while the speed is between the limits, but the torque demand output is constrained to lie on the specified torque-speed curve.

In addition to the **Optimal mode gain**, the **Optimal mode maximum speed** must be specified. This must be no greater than the **Maximum generator speed.** The controller will adjust torque demand as above to maintain the desired tip speed ratio provided the generator speed lies between the **Minimum generator speed** and the **Optimal mode maximum speed**. If either of these speed limits is reached, it is used as the speed set-point for the Speed Control by Torque Demand PI controller[5.5]. The deviation of measured generator speed from this set-point is the input to the controller, and generator torque demand is the output. Thus the generator torque is varied to keep the generator speed at the limit.

Note: This PI controller remains active while the speed is between the limits, but the torque demand output is constrained to lie on the quadratic curve defined by the **Optimal mode gain**.

Note also that energy output may not be maximised by maximising aerodynamic efficiency, because the energy losses[4.9] may also vary with the operating point. It may therefore be better to track a slightly different tip speed ratio. The desired tip speed ratio and the corresponding power coefficient should be used in calculating the **Optimal mode gain** as above.

Further details are given in the Theory Manual.

5.7.4 Control in the variable slip case

With a variable slip generator, it is sufficient to set a high torque demand below rated: the generator itself will restrict operation to the nominal slip curve. This can easily be specified using the look-up table option - see Section 5.7.2.

5.8 User-defined controllers

Although ***Bladed*** has a full set of built-in controllers, both for power production control and for supervisory control, a wide range of different control algorithms is used in practice by different turbine manufacturers. The controller details can have a profound effect on turbine loads and performance, so it is important to allow the flexibility to use any control logic the user wishes to define. Also it is worth considering whether the same code can be used for controlling ***Bladed*** simulations as is used in the actual turbine controller.

Bladed allows a user-defined controller to be used for any of the following tasks:

- Control of blade pitch and generator torque across the whole range of operation, including power production, normal and emergency stops, starts, idling and parked conditions.
- Control of the shaft brake and generator contactors.
- Control of nacelle yaw.

You may use a mixture of user-defined and built-in controllers for the various different controller functions.

Note that the user-defined controller operates on a discrete timestep, like most real controllers. The built-in controllers operate in continuous time, but can be used to approximate the behaviour of a discrete controller as long as its timestep is not too long.

The user-defined controller may be written in any language, either as a DOS or Windows executable program (.exe) capable of reading and writing to shared files, or as a 32-bit DLL (dynamic link library). A DLL is preferable as it will result in faster simulations, and communication with ***Bladed*** may be more reliable.

5.8.1 Writing a user-defined controller as an executable program

If the controller is written as an executable, it will use a shared file for two-way communication with ***Bladed***. When the simulation starts up, the controller executable (.exe) file is first copied into the directory where ***Bladed*** is installed, and renamed **discon.exe**. When **discon.exe** starts up, the ***Bladed*** directory is the current directory, and the files used to communicate between the simulation and the controller are in this directory. The controller program can therefore refer to these files by name without giving the full path.

Two files are used to communicate between the user-defined controller program and the simulation. One of these is a text file named **discon.aux**, which is written by the simulation and just contains the directory and run name for the simulation results. This may be useful if the controller wishes to write any permanent record of what it does to be stored with the simulation results. The file has just two lines: the first consists of the word **PATH** followed by a space and then the path for the simulation results (including the final backslash). The second line consists of the word **RUNNAME** followed by a space and then the run name, i.e. up to 8 characters. The controller may choose to ignore this file if the information is not required.

The second file is used for the dynamic information exchange between the two programs. It is called **discon.swp**, and is a binary file with a record length of 4 bytes. The file must be opened as a shared file, allowing simultaneous read and write access to both programs. The file structure is given in Appendix A.

Although this file has many records, it may only be necessary for the external controller to read from and write to a small number of these, depending on the turbine type and the tasks which the external controller is performing.

Handshaking: record 1 of **discon.swp** is used for handshaking, to ensure that neither program starts reading data until the other program has finished writing it. The sequence of events to be followed by the controller program is as follows:

1. Controller program starts by creating **discon.swp** and writing a zero to record 1.
2. Controller program waits until record 1 becomes 1 or -1. If it is 1, this indicates that the simulation has finished writing data, and also that the file **discon.aux** is ready if required. If it is -1, the simulation is about to finish and the controller program should stop.
3. If the first record is 1, the controller may read any of the parameters written by the simulation, perform its calculations, and then write the appropriate outputs. Once all data is written, the controller writes a zero to record 1 to tell the simulation that it is ready. If the controller decides to abort the simulation, it should write -1 to record 1, and write an appropriate message to **discon.swp** as described in Appendix A.
4. Controller returns to step 2.

In step 2, the controller waits until record 1 becomes non-zero. It is important that the controller closes and re-opens **discon.swp** every time around the loop, otherwise the contents of the file as represented in the disk cache may not have been updated.

5.8.2 Writing a user-defined controller as a dynamic link library

A dynamic link library provides faster and more reliable communications between the controller and the ***Bladed*** simulation, and is recommended. The interface to ***Bladed*** is also simpler to write.

When the simulation starts up, the controller DLL file is first copied into the directory where ***Bladed*** is installed, and renamed **discon.dll**. When **discon.dll** is first called, the ***Bladed*** directory is the current directory.

The controller is written as a subroutine or procedure. The DLL export name of the procedure must evaluate to DISCON (note: this name must be in upper case). Depending on the language system being used, it may be necessary to define this by means of an alias. The procedure does not generate a return value. It has five arguments, as follows (the names given here are arbitrary, and are given purely for ease of reference within this manual. Only the order is important):

"DATA" The address of the first record of an array of single-precision (4-byte) real numbers which is used for data exchange between the simulation and the controller. The contents of the array is given in Appendix A.

"FLAG" A 4-byte integer (passed by reference) which the DLL should set as follows:
0 if the DLL call was successful
>0 if the DLL call was successful but the "MESSAGE" should be issued as a warning message. The simulation will continue.

<0 if the DLL call was unsuccessful or for any other reason the simulation is to be stopped at this point. “MESSAGE” is then issued as an error message.

“INFILE” The address of the first record of an array of 1-byte characters giving the name of the parameter file, which is currently DISCON.IN (See section 5.8.3). This array should not be modified by the DLL. The number of characters in the name is given in “DATA” - see Appendix A.

“OUTNAME” The address of the first record of an array of 1-byte characters giving the simulation run name, prefixed by the full path to the directory which will contain the simulation results. This may be useful if the controller wishes to write a permanent record of what it does to be stored with the simulation results: the results should be stored in a file whose name (including path) is generated by appending “.xxx” to “OUTNAME”, where xxx is any suitable file extension **not** beginning with “%”. The number of characters in the name is given in “DATA” - see Appendix A. Alternatively, or in addition, the DLL may send information back to Bladed for output in the same form as the other simulation results. This is described in Appendix A.

“MESSAGE” The address of the first record of an array of 1-byte characters which may be used by the DLL to send a text message to ***Bladed***, which appears on the screen and is stored together with any other calculation messages generated by ***Bladed***.

5.8.3 Using a user-defined controller

To use your own controller for any of the control functions, select the External Controller option where appropriate on the Control Systems or Yaw Control windows.

To specify your controller, select the External Controller tab on the **Supervisory Control** window; on the Control Systems window, **Define...** buttons are provided for this in appropriate places. Then specify the following information:

Controller code: the compiled .exe or .dll file which runs your controller (a dialogue box allows you to select the appropriate file)
Communication Interval: the controller timestep. At these regular intervals, ***Bladed*** will send information to the external controller, and expect to receive information in return. In real life the controller will take some time to calculate its outputs, usually a whole timestep, so it may be appropriate for the controller to always return the results of the previous timestep’s calculation.
Noise on measured signals: if desired, click **Specify noise** to specify any random noise and discretisation errors[5.8.4] on the signals passed to the external controller.
External Controller Data (optional)**:** The information you provide here will be placed in a text file DISCON.IN which your controller is free to open and read when it starts up (no path is required as the file will be in the ***Bladed*** directory, which is the current directory when the external controller starts. You can use this to pass additional parameters to your controller, e.g. gains, look-up tables, etc. There is no need to pass the set-points etc. which are defined in the steady-state part of the Control Systems window, as these will be passed across in the communication file or array. (Note: to enter a tab in this field, press Control+Tab).

Important note:
When using a user-defined controller, ***Bladed*** evidently cannot guarantee that the performance defined by the steady-state control parameters will actually be achieved on average during a dynamic simulation.

5.8.4 Signal noise and discretisation

When using an external controller, click **Noise on measured signals** and **Specify noise** to specify random noise and discretisation errors on the measured signals passed to the external controller. For each signal, select the type of random noise (none, rectangular distribution or Gaussian distribution). For a rectangular distribution, specify the half-width of the distribution (i.e. the maximum error). For a Gaussian distribution specify the standard deviation. The discretisation step for each signal may also be specified – the signal is discretised after any random noise has been added.

5.9 Transducers

Transducer models are available to provide the following measured signals:

- Electrical power
- Generator speed

Each transducer is modelled as a first order lag. Set a zero time constant to ignore transducer dynamics.

The electrical power signal is required for the in-built dynamic fixed speed pitch regulated controller. The generator speed signal is required for the in-built dynamic variable speed controllers. Both signals are available to the external controller if required.

The electrical power is at the turbine terminals, i.e. net of any electrical losses including auxiliaries.

5.10 The pitch actuator

The dynamics of the pitch actuator are an important part of the control loop dynamics for pitch controlled turbines, and must be defined before dynamic simulations can be carried out for these turbines. Click **Control** on the main toolbar followed by **Pitch Actuator**, or use the **Specify -> Control systems** pull-down menu. (For aileron or air-brake control, the pitch actuator is actually an aileron or air-brake actuator.)

Simple passive actuator models are provided, and are suitable for many calculations. More complex models are useful for detailed calculations, particularly if pitch actuator loads and detailed behaviour need to be simulated. Proceed as follows:

Input signal: select whether the command received from the controller is a pitch position demand or a pitch rate demand.
Individual pitch control: Select this if the user-defined controller[5.8] will generate separate pitch demand signals for each blade. Selecting this option will also cause the pitch actuators

on each blade to be modelled individually, so that the loads for each will be output separately. Otherwise only the mean loads across all the actuators will be output.

Response to pitch position demand: if the input signal is a pitch position demand, specify the actuator response as **Passive** (1st, 2nd or higher order models are provided) *5.10.1* or **Active**, and click **Define...** to specify the appropriate parameters. In the **Active** case, a PID controller on pitch error generates a pitch rate demand*5.10.2*. An optional **Ramp Control** is provided, which converts step changes in raw position demand coming from the discrete controller into a smooth profile based on the specified rate and/or acceleration limits.

Response to pitch rate demand: if the input signal is a pitch rate demand, or if it is pitch position but the response to is **Active**, then the response to the pitch rate demand muct be specified. This may be **Passive** (1st, 2nd or higher order models are provided)*5.10.1* or **Active.** Click **Define...** to specify the appropriate parameters. In the **Active** case, a PID controller on pitch rate error generates an actuator torque demand*5.10.2*.

Pitch position limits (hardware): pitch actuator hard stops must be specified in the case of pitch rate demand input, otherwise they are optional.
Pitch rate limits (hardware): any actuator rate limits may be specified if required.

Actuator, blade pitching inertia and pitch bearing details:
For passive actuator models, this information is optional and is only used to generate additional calculation outputs such as the pitch bearing friction and pitching inertia (note that this varies with blade bending).

If **Actuator details** are supplied for the case of **passive response to pitch rate demand**, then the response will be affected by the acceleration limits calculated from the actuator torque limits, the applied pitching moment and any bearing friction. This is a useful model for hydraulic actuation systems where the flow rate to the cylinder is controlled by a proportional valve. The gearbox ratio and torque limits will need to be calculated by treating the linear actuator and pitch linkage as if it were a rotational actuator. This model can also be used to give a simplified representation of electrical actuators.

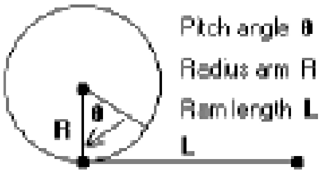
If **Active** response to pitch rate demand is specified, then the actuator and bearing details must be supplied. The response of the actuator to a torque demand is represented by a passive 1st, 2nd or higher order response generating an actual torque*5.10.1*. Click **Define...** to enter the appropriate parameters. For an instantaneous response, select a first order model with zero time constant. The resulting torque is subject to limits as specified. This model is useful for electric actuation systems, but could also be used for hydraulic systems operated by direct control of the hydraulic pressure.

Select a **rotary** or **linear** actuator as appropriate. The rotary model is particularly useful for electrical actuators, while the linear model is useful for hydraulic cylinders. The rotary model requires the following parameters:

Gearbox ratio: the step-down ratio from pitch motor speed to pitch rate.
Gearbox efficiency: this represents the torque loss in the gearbox.
Rotational inertia: the motor inertia (referred to its own shaft).

For the linear actuator model, the gearbox ratio would normally be 1.0, and the effective inertia is likely to be small enough to neglect. The actuator torque is defined for the maximum torque position of the actuator (when the ram is tangential to the pitch circle).

Click the **Geometry** button to define the corresponding pitch angle and the actuator length (between pivots) when in this position, and also the radius arm. The actuator force limits are assumed constant, so at other pitch angles the torque limits will be reduced.



Linear actuator geometry at the maximum torque position. If the pitch angle decreases with increasing ram length, set the ram length L to be negative.

Single actuator: the parameters supplied relate to a single actuator controlling all the blades together.
One actuator per blade: there is a separate actuator for each blade, and the parameters supplied relate to each actuator. This option must be selected if individual pitch control is to be used.

Blade pitching inertia: the inertia about the pitch axis for a single blade (undeflected).
Bearing friction: the friction torque (on each bearing) is the sum of a constant term, a term proportional to the applied bending moment, and a term proportional to the applied axial force on the bearing, i.e. the radial force acting outwards along the pitch axis, and a term proportional to the radial force on the bearing, i.e. the magnitude of the vector sum of the blade in-plane and out of plane shear forces at the bearing position.
Bearing 'stiction': when stationary, the break-out torque is equal to the bearing friction torque described above plus an additional stiction torque. This stiction torque is represented by a constant torque plus a term proportional to the friction torque.

5.10.1 Passive dynamics

Click the appropriate **Define...** button and enter the following information:

First order passive The pitch actuator is modelled as a first order lag. Enter the time constant.

or

Second order passive Enter the natural frequency and the fraction of critical damping for the second-order transfer function

or

Other passive Enter the required coefficients for powers of the Laplace variable s in the transfer function numerator and denominator. The numerator must have lower order than the denominator.

5.10.2 PID parameters

Click the appropriate **Define...** button and enter the following information for the closed loop PID controller which generates either a pitch rate demand output signal from a pitch position error input signal, or a torque demand output signal from a pitch rate error input signal.

Then if the input error signal is $x = x_d - x_m$ (x_d is the demand, and x_m the measured or feedback signal) and the output demand signal is y, the Laplace representation of the PID controller is:

$$y = \left(K_p + \frac{K_i}{s}\right)x + \frac{K_d s}{1 + sT_d} x_1$$

Enter the following parameters:

Input filter time constant: a 1st order pre-filter which smooths step changes in demand coming from a discrete controller. Specify zero for no pre-filter.

Proportional gain: K_p

Integral gain: K_i

Differential gain: K_d

Applied to: The differential term may apply to the feedback ($x_1 = x_m$), demand ($x_1 = x_d$) or error ($x_1 = x$) term - see below. Specify $x_1 = x_d$ to give feed-forward action.

Filter time constant: T_d (prevents excessive high frequency gain of the differential term).

Additional detail on the pitch actuator model can be found in the ***Bladed*** theory manual.

5.11 The shaft brake

Click the **Brakes** button, either on the **Transmission** screen or in the Supervisory Control section of the **Control Systems** screen, to define the dynamic characteristics of the shaft brake.

The mechanical shaft brake is used for simulations of stopping or parked rotors. When the brake is applied, it usually takes a little time to reach full torque. Two ways are provided for specifying how the brake torque develops. Select **Linear Ramp** or **Look-up Table** by clicking the appropriate button:

Linear ramp model

The brake torque increases linearly from zero up to full braking torque. Enter the following data in the boxes provided:

- **Maximum shaft brake torque**: the full braking torque, and
- **Shaft brake ramp time**: the time to reach full torque starting from zero.

Look-up table

The braking torque may be specified as a time history of braking torque against time since the brake application was initiated. Use the **Add** button to add points to the time history.

See Drive train[4.6.2] to specify the position of the brake.

See also: Normal stop sequence[5.13], Emergency stop sequence[5.14], Parked conditions[5.16].

5.12 Start-up sequence

For simulations of start-up, click the **Start** button on the **Control** screen and specify the following parameters:

- **Initial rotor speed**: the rotor speed at the start of the simulation.
- **Initial pitch angle**: the pitch angle at the start of the simulation.

The built-in start-up logic may be used, in which case the following parameters should be entered to define the sequence of events:

- **Initial pitch rate during start-up**: the constant pitch rate which will be used during the start-up sequence.
- **Generator speed at which generator is put on line**: As soon as the generator speed reaches this value, the generator is connected to the grid and the power production control[5] starts to act. However any pitch angle demanded by the controller will be ignored until the initial pitch ramp is complete, as defined by:
- **Final pitch angle in start-up mode**: The **Initial pitch rate during start-up** continues to act until this pitch angle is reached, at which point any pitch angle demanded by the power production controller starts to be used.

Alternatively, click **Defined by external controller** to use your own start-up logic.

Notes:
1. The sign of the Initial pitch rate must be consistent with the initial and final pitch angles: if the final pitch angle is less than the initial pitch angle, the pitch rate must be negative, and vice versa.
2. Pitch angles and rates apply also to aileron/flap/airbrake deployment angles. They are ignored if the rotor has no aerodynamic control surfaces[4.1].

5.13 Normal stop sequence

Normal stops use aerodynamic braking and/or the shaft brake to stop the rotor from the normal running condition. For simulations of normal stops, click the **Normal stop** button on the **Control** screen. To use the built-in logic for normal stops, specify the following parameters:

- **Pitch rate**: the constant pitch change rate used during the stop sequence.
- **Maximum pitch**: the final pitch angle. If the Pitch rate is negative, the Maximum pitch should actually be the most negative pitch angle. The initial pitch angle will depend on the running conditions before the stop occurs.
- **Rotor speed for cut-in of shaft brake**: when the rotor has slowed down to this value, the shaft brake[5.11] is applied to bring the rotor to rest.

The normal stop begins at the **Time to begin a stop** specified in Simulation control[7.13] .

Alternatively, click **Defined by external controller** to use your own normal stop logic. Note that the **Time to begin a stop**[7.13] does not apply in this case.

Notes:
1. Pitch angles and rates apply also to aileron/flap/airbrake deployment angles. They are ignored if the rotor has no aerodynamic control surfaces[4.1].
2. User-defined supervisory control may also be defined in an external controller[5.8].

5.14 Emergency stop sequence

Emergency stops use aerodynamic braking and/or the shaft brake to stop the rotor in the event of a loss of load. Either braking mechanism may be tripped either by the loss of load or by a specified overspeed condition.

For simulations of emergency stops, click the **Emergency stop** button on the **Control** screen. To use the built-in logic for emergency stops, specify the following parameters:

- **Emergency pitch trip mode**: Select Overspeed if aerodynamic braking is to be triggered by an overspeed, or Grid loss if it is to be triggered immediately on loss of load.
- **Rotor speed at which pitch motion commences**: The overspeed condition which triggers the aerodynamic braking if Emergency pitch trip mode is set to Overspeed.
- **Emergency pitch rate**: the constant pitch change rate which is used following the Grid loss or Overspeed.
- **Maximum pitch**: the final pitch angle. If the Emergency pitch rate is negative, the Maximum pitch should actually be the most negative pitch angle. The initial pitch angle will depend on the running conditions before the stop occurs.
- **Emergency shaft brake trip mode**: Select Overspeed if the shaft brake is to be triggered by an overspeed, or Grid loss if it is to be triggered immediately on loss of load.
- **Rotor speed for cut-in of brake (speed control)**: The overspeed condition which triggers the shaft brake if Emergency shaft brake trip mode is set to Overspeed.
- **Rotor speed for cut-in of brake (for parking)**: when the rotor has slowed down to this value, the shaft brake is applied to bring the rotor to rest if it has not already been applied as a result of grid loss or overspeed..

The emergency stop begins at the **Time to begin a stop** specified in Simulation control[7.13] .

Alternatively, click **Defined by external controller** to use your own emergency stop logic. Note that the **Time to begin a stop**[7.13] does not apply in this case.

Notes:
1. Pitch angles and rates apply also to aileron/flap/airbrake deployment angles. They are ignored if the rotor has no aerodynamic control surfaces[4.1].
2. User-defined supervisory control may also be defined in an external controller[5.8].

5.15 Idling conditions

For simulations of an idling rotor, click the **Idling** button on the **Control** screen and specify the fixed **pitch or aileron angle** to be used while idling. Click **Enable external controller** if any user-defined control logic is to be used.

5.16 Parked conditions

For simulations of a parked rotor, click the **Parked** button on the **Control** screen and enter:

- the fixed **pitch or aileron angle** to be used when parked, and
- the **rotor position** or azimuth angle when parked (note that this position is used at the start of the simulation, but it is possible for the brake to slip, allowing the azimuth to change during the simulation). A zero azimuth angle means that the first blade is pointing vertically upwards.

Click **Enable external controller** if any user-defined control logic is to be used.

5.17 Yaw control

5.17.1 Yaw Dynamics

Four yaw dynamics options are available. Click the **Yaw control** button on the **Control Systems** window and select the appropriate option:
None: the nacelle direction will remain fixed.
Rigid yaw: the nacelle follows the demanded nacelle angle*5.17.2* precisely.
Flexible yaw: the nacelle yaws passively with respect to the demanded nacelle angle*5.17.2* as a result of aerodynamic forces.
Controlled torque: the yaw actuator torque is specified by the external controller*5.17.2*.

If **Flexible Yaw** or **Controlled torque** is selected, you may specify a
Yaw friction torque and any
Additional stiction. The nacelle will only start to yaw if the balance of applied yawing torques at the yaw bearing bearing exceeds the friction plus the stiction torque, and it will stop yawing if the balance of torques falls below the friction torque. The applead torques include aerodynamic and inertial Mz torques and either the spring/damper torque (in the case of **Flexible yaw**) or the **Controlled torque**.

If **Flexible Yaw** is selected, specify the:
Yaw Damping (if any), and either the
Yaw Stiffness (if any) for a linear spring model, or select the hydraulic accumulator model.

The following parameters are required to model yaw compliance provided by a hydraulic accumulator system:
Gas volume (V): the volume of gas in the accumulators
Nominal pressure (P): the equilibrium pressure in the hydraulic system
Pump flow per unit yaw (F): the volume of fluid which must pass through the yaw motor to achieve a one radian change in nacelle angle
Yaw torque per unit pressure (Q): the relationship between the pressure difference across the yaw motor and the torque developed at the yaw bearing
Gas law constant: the constant γ for the gas in the accumulator. In the gas law equation,

$$PV^{\gamma} = RT$$

Specify $\gamma = 1$ for isothermal conditions.

The yaw torque **Y** provided by the hydraulic system is then given by:

$$Y = Q.P\left(\left[\frac{V}{v_1}\right]^{\gamma} - \left[\frac{V}{v_2}\right]^{\gamma}\right)$$

where $v_1 = V - F.\alpha$, $v_2 = V + F.\alpha$, and α is the difference between the actual and the demanded nacelle angle.

5.17.2 Active Yaw

Three choices are available for defining the demanded nacelle angle in simulations:

- **None**: the demanded nacelle angle will be fixed at zero (North).
- **Prescribed manoeuvre**: specify a yaw manoeuvre by entering:
    - **Time to start yaw manoeuvre**: when the demanded nacelle angle starts to move
    - **Required yaw position change**: the angle through which it will move
    - **Yaw rate for yaw manoeuvres**: the yaw rate to be used.
- **External controller**: if **Rigid yaw** or **Flexible yaw** is selected[5.17.1], the user-defined controller will generate a yaw rate demand which defines the **demanded nacelle angle** at any instant. If **Controlled torque** is selected, the user-defined controller[5.8] will generate a yaw actuator torque demand. This will be the same as the actual yaw actuator torque, since the dynamic response of the yaw torque actuator is assumed to be fast, and is therefore not represented in the model.

6. DEFINING THE ENVIRONMENT

This section describes how to define the environmental conditions in which the turbine operates. The environmental conditions include wind (Section 6.1), waves (Section 6.12) and currents (Section 6.13).

6.1 Defining the wind

Click the **Wind** icon on the toolbar to define the characteristics of the wind field.

For the calculation of steady state parked loads and for all simulations, define the steady-state spatial distribution of the wind. For simulations, define also the time-varying wind characteristics.

The steady-state characteristics of the wind field may include any combination of the following elements:

- Wind shear[6.2] : the variation of wind speed with height.
- Tower shadow[6.3] : distortion of the wind flow by the tower.
- Upwind turbine wake[6.4] : full or partial immersion in the wake of an upstream turbine.

For simulations, wind speed may vary with time in addition to the steady-state spatial characteristics defined above. Click on **Time Varying Wind** and choose one of the following options:

- No variation[6.5] : for simulations in which the wind speed and direction are constant in time.
- Single point history[6.6] : to supply a time history of wind speed and direction which is fully coherent over the whole rotor.
- 3D turbulent wind[6.7] : use a 3-dimensional turbulent wind field with defined spectral and spatial coherence characteristics. See Generating turbulent wind fields.
- Transients[6.8] : to use sinusoidal wind speed and direction transients as defined by certain standards, such as IEC 1400-1.

These options are described below, starting at Section 6.5. In each case the wind characteristics are defined at a specified height, and the wind speed at other heights will be modified according to the shear profile. Click **Refer wind speed to hub height** to ensure that the definitions always apply at the hub height of the turbine.

Click on **View wind data** for a graphical representation of the wind field. The options available include:

- Time history plots for selected positions in the rotor plane
- 3D carpet plots and animations showing the wind field over the whole rotor plane

The variables which may be displayed include:

- Longitudinal, lateral and vertical wind speeds
- Resultant horizontal wind speed

- Total resultant wind speed
- Wind direction and upflow angle.

This data may also be tabulated as for other graphical data[9.9].

The time history plots optionally include moving-averaged values if a gust average period is specified. Click **Stats** to see the mean wind speed and the maximum and minimum values of the wind speed and the moving-average gust wind speed. Click **Max Gust Table** to view a table of maximum moving-average gust values at a grid of points, after first specifying the spacing of the grid points required. Note that this calculation may be quite slow.

6.2 Wind shear

Wind shear is the variation of steady state mean wind speed with height. If wind shear is required, click the **Wind** icon on the toolbar and select **Wind shear.** Then select either the exponential or the logarithmic wind shear model.

Exponential model:

Enter the wind shear exponent. The wind speed V(h) at height h above the ground is then given by:

$$V(h) = V(h_0)\left(\frac{h}{h_0}\right)^{\alpha}$$

where

h_0 is the reference height, and

α is the wind shear exponent.

A zero exponent results in no wind speed variation with height.

Logarithmic model:

Enter the ground roughness height parameter. The wind speed V(h) at height h above the ground is then given by:

$$V(h) = V(h_0)\left(\frac{\log(h / z_0)}{\log(h_0 / z_0)}\right)$$

where

h_0 is the reference height, and

z_0 is the ground roughness height.

6.3 Tower shadow

Tower shadow defines the distortion of the steady-state mean wind field due to the presence of the tower. Click the **Wind** icon on the toolbar and select **Tower shadow**, and select a model.

Potential flow model:

This model is appropriate for rotors operating upwind of the tower. Enter the **Tower diameter correction** factor, *F*. The longitudinal wind velocity component around the tower is modified using the assumption of incompressible inviscid flow around a cylinder of diameter *F.D*, where *D* is the tower diameter at the height where the tower shadow is being calculated.

Empirical model:
For rotors operating downwind of the tower, use this empirical model, which uses a cosine bell-shaped tower wake. Enter the following parameters:

- **Maximum velocity deficit** at the centre of the wake as a fraction of the local wind speed.
- **Width of the tower shadow** as a fraction of the local tower diameter.
- **Reference position**: the distance downwind, as a proportion of the local tower diameter, at which the above parameters are defined. At other distances, the shadow width increases and the velocity deficit decreases with the square root of the distance from the tower.

Combined model:
This model uses the potential flow model at the front and sides of the tower, and the empirical model on the downwind side wherever it gives a deficit greater than the potential flow model deficit, with a smooth transition between the two. This is useful in simulations where the rotor might yaw in and out of the downwind shadow area.

6.4 Upwind turbine wake

If the turbine rotor being modelled (the target turbine) is wholly or partially immersed in the wake of another turbine operating further upwind, use this option to define the modification to the steady-state mean wind profile caused by the wake of the upwind turbine.

Click the **Wind** icon on the toolbar and select **Upwind turbine wake**, and choose between the simple Gaussian wake profile and the Eddy viscosity model.

For the simple Gaussian model, two parameters define the wake profile:
- **Centre line velocity deficit**: as a percentage of the local wind speed.
- **Wake half-width**: the distance from the wake centre line at which the deficit is reduced to $e^{-0.5}$ times the centre line value.

The eddy viscosity model carries out a full calculation of the wake, based on the following characteristics of the upwind turbine:

- **Diameter**
- **Thrust coefficient**
- **Tip speed ratio**
- **Number of blades**
- **Distance upstream** (from the target turbine).

With this model, the wake development depends on the ambient turbulence intensity. For turbulent wind simulations the ambient turbulence is defined by the time varying wind[6.7] definition. Enter a default turbulence intensisty for use in other calculations.

The eddy viscosity model also includes a calculation of the added turbulence generated by the wake. For turbulent wind simulations, the ambient turbulence defined in time varying wind[6.7] will be increased accordingly. A choice is provided to control how the added turbulence is interpreted for the lateral and vertical components of turbulence:

- **Not calculated**: The added turbulence is ignored.
- **Longitudinal only**: the longitudinal turbulence intensity is increased using the calculated added turbulence, but the lateral and vertical turbulence intensities are not increased.
- **Longitudinal; others pro rata**: the lateral and vertical turbulence intensities are increased in the same proportion as the longitudinal turbulence intensity.
- **Applied to all components** (default): the variance of each component is increased by the same amount, i.e. by the variance of the added turbulence.

For both wake models, the wake centreline may also be offset from hub position of the target turbine by entering:

- **Horizontal offset**: the horizontal distance between the rotor centre and the centre line of the wake. A positive value indicates that the wake centre passes to the right of the rotor centre when looking in the direction of the wind flow.
- **Vertical offset**: the vertical distance between the rotor centre and the centre line of the wake. A positive value indicates that the wake centreline is above the target turbine hub.

6.5 No time variation of wind speed

Use this option to specify the wind conditions for a simulation when no variation of the wind field with time is required. Any steady-state spatial variations defined by wind shear, tower shadow and an upwind turbine wake will apply.

Click the **Wind** icon on the toolbar, select **Time varying wind**, and choose the **No variation** option**.**

Enter the following parameters:

- **Wind speed**: the steady wind speed to be used in the simulation.
- **Height at which speed is defined**: the reference height to which the **wind speed** applies, unless **Refer wind speed to hub height** has been selected. If wind shear[6.2] is defined, the wind speed at any other height will be different.
- **Wind direction**: measured clockwise from North - see wind direction[6.9].
- **Flow inclination**: for non-horizontal wind flows (e.g. on the side of a hill). A positive value indicates a rising wind.

6.6 Single point wind history

Use this option to specify the wind conditions for a simulation when an arbitrary time-varying wind speed and/or direction is required, but no spatial variation is required other than the steady-state characteristics defined by wind shear, tower shadow and an upwind turbine wake.

Click the **Wind** icon on the toolbar, select **Time varying wind**, and choose the **Single point history** option.

Enter the following parameters:

- **Height to which speeds relate**: the reference height to which the wind speed time history applies, unless **Refer wind speed to hub height** has been selected. If wind shear[6.2] is defined, the wind speed at any other height will be different.
- **Flow inclination**: for non-horizontal wind flows (e.g. on the side of a hill). A positive value indicates a rising wind.

Then use the **Add** button to add points to the time history. For each point, enter

- **Time**: this should start from zero.
- **Speed**: the wind speed at that time.
- **Wind direction**: measured clockwise from North - see wind direction[6.9].

6.7 3D turbulent wind

Use this option to perform a simulation in which the turbine is immersed in a turbulent wind field which varies both in space and time. This turbulent wind field is superimposed on any steady-state spatial variations defined by wind shear, tower shadow and an upwind turbine wake. The turbulent variations at any point in the rotor disk have defined spectral characteristics, and the variations at any two points in the rotor disk are correlated by a defined coherence relationship representative of the spatial structure of real atmospheric turbulence. The turbulent wind field may have just the longitudinal component of turbulence, or it may have all three components.

Click the **Wind** icon on the toolbar, select **Time varying wind**, and choose the **3D turbulent wind** option.

Enter the following information:

- **Turbulent wind file name**: Click here to select a file which contains an appropriate turbulent wind field: see Generating turbulent wind fields[6.10]. The characteristics of the turbulent wind field will be displayed as you select a file. The wind field should be high enough and wide enough to envelop the whole rotor, and long enough to allow the simulation to continue for as long as is required. If U is the average wind speed corrected to hub height according to the wind shear, and the simulation is to run for a time T, the wind field length should be $U.T$ + Turbine diameter. It may be longer than this, although in this case the mean wind speed and turbulence intensity for the simulation may not match the values selected, as they apply only if the whole file is used.
- **Mean wind speed**: the mean wind speed for the turbulent wind at the reference height. If this does not match the wind speed for which the turbulence characteristics were defined, the turbulence field will be scaled appropriately. However, since the dimensionless characteristics of turbulence are not quite invariant with wind speed, this scaling is not strictly valid.

- **Height at which speed is defined**: the reference height to which the **Mean wind speed** applies, unless **Refer wind speed to hub height** has been selected. If wind shear[6.2] is defined, the mean wind speed at any other height will be different.
- **Turbulence intensity**: the standard deviation of turbulent wind speed variations as a fraction of the mean wind speed. If using a three-component wind field, specify the turbulence intensity for each component.
- **Wind direction**: measured clockwise from North - see wind direction.
- **Flow inclination**: for non-horizontal wind flows (e.g. on the side of a hill). A positive value indicates a rising wind.

For the **Height of turbulent wind field**, select one of the following options. These do not affect the mean wind speed at any height, only the location of the turbulent wind fluctuations:

- **Centred on hub height**: The centre of the turbulent wind field is placed at hub height.
- **Best fit for rotor and tower**: If the wind field vertical dimension is less than the height of the turbine, the top of the wind field will be located at the height of the top of the rotor, so that the wind field extends as far down the tower as possible. If the wind field vertical dimension is greater than the turbine height, the wind field will start at ground level and will envelop the whole turbine.

Turbulent variations below the bottom of the defined wind field will be taken from the lowest defined point.

The wind turbulence is defined at a number of grid points[0], both the the rotor plane and in the alongwind direction. In between these points, either linear or cubic interpolation can be used to determine the turbulence. Specify one of the following options for the interpolation scheme:

- **Linear**: uses linear interpolation
- **Cubic in rotor plane only**: uses cubic interpolation laterally and vertically, and linear interpolation in the alongwind direction
- **Fully cubic**: uses three-dimensional cubic interpolation.

If desired, specify also a superimposed sinusoidal wind direction transient. This is particularly useful if only a single turbulence component is being used. Enter the following parameters:

- **Amplitude of direction change**: the peak-to peak amplitude of the transient. Note that the specified wind direction (see above) refers to the start of the simulation. The mean wind direction will be different from this if a transient is added.
- **Start time of transient**: the time into the simulation at which the transient starts.
- **Duration of transient**: the duration of the transient from the time it starts to the time it finishes.
- **Type of transient**: half or full wave: see Section 6.8.

6.8 Transients

Use this option to define sinusoidal wind speed and direction transients such as are defined in certain standards, such as, for example, IEC 1400-1. These transients are superimposed on any steady-state spatial variations defined by wind shear, tower shadow and an upwind turbine wake.

Click the **Wind** icon on the toolbar, select **Time varying wind**, and choose the **Transients** option.

First enter the following data:

- **Reference height** at which the wind speed is defined, unless **Refer wind speed to hub height** has been selected. If wind shear[6.2] is defined, the wind speed at any other height will be different.
- **Flow inclination**: for non-horizontal wind flows (e.g. on the side of a hill). A positive value indicates a rising wind.

Separate transients may be defined for each of the following variables:

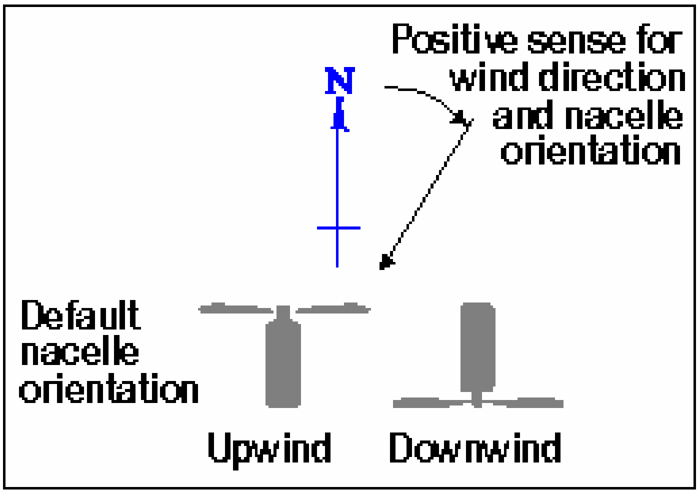
- **Wind speed**: defined at the **reference height**.
- **Wind direction**: measured clockwise from North - see wind direction[6.9].
- **Horizontal wind shear**: defined as the difference in wind speed between the two sides of the rotor plane. The wind speed varies linearly with horizontal distance across the rotor plane.
- **Vertical wind shear**: defined as the difference in wind speed between the top and bottom of the rotor plane. The wind speed varies linearly with height.

All four transients will be used, but if any is not required, specify an appropriate start value and a zero amplitude. For each transient, enter the following information for the variable in question:

- **Start value**: the value of the variable before the start of the transient.
- **Amplitude of change**: the largest range from maximum to minimum during the transient.
- **Time to start cycle**: the time into the simulation at which the transient starts.
- **Time period of cycle**: the duration of the transient.
- **Type of cycle**: Three types of transient are provided, as specified by various standards. A **half wave** transient begins with the start value and increases sinusoidally by the amount defined by the Amplitude. A **full wave** transient begins with the start value and changes sinusoidally by the amount defined by the Amplitude in the first half of the time period, changing back to the start value by the end of the time period. The **IEC-2** transient is a more complex shape, as described in the Theory Manual.

6.9 Definition of wind direction

The default nacelle orientation and the wind direction together define the nacelle yaw angle. This may change during a simulation if any yaw control[5.17] is specified, i.e. either passive yaw or a yaw manoeuvre. Note that the yaw angle can also be defined explicitly for the steady parked loads[7.10] calculation.



6.10 Generating turbulent wind fields

In order to run a simulation which uses a 3-dimensional turbulent wind field, a suitable wind field must first be generated. Once generated, these turbulent wind fields are stored so that they can be used for future simulations.

Turbulent wind fields contain time histories of wind speed at each of a number of points on a rectangular grid covering the turbine rotor swept area. The time history at each point will have spectral characteristics conforming to either the von Karman or the Kaimal model of wind turbulence. The time histories at any two points in the rotor plane will be correlated with each other in accordance with an appropriate model of the lateral and vertical coherence characteristics of atmospheric turbulence. Linear interpolation is used between grid points, as well as between successive time points.

To generate a new wind field, first define the turbulence characteristics and then generate the wind field as described below.

6.10.1 Defining turbulence characteristics

Click the **Wind** icon on the toolbar and select **Define turbulence**. Then define following:

- **Spectrum type**: select either von Karman or Kaimal to generate just the longitudinal component of turbulence, or 3-component von Karman if all three components are required.
- **Number of points along Y**: the number of grid points horizontally.
- **Number of points along Z**: the number of grid points vertically.
- **Length of Y**: the width of the turbulent wind field - this should be at least the rotor diameter, or the greatest distance from the tower centreline to the blade tip to allow for operation at non-zero yaw angles.
- **Length of Z**: the height of the turbulent wind field - this should be at least the rotor diameter.
- **Length of X**: the length of the turbulent wind field - this should be sufficient for the length of simulation required, at the appropriate wind speed. The 'frozen' wind field moves past the turbine at the specified wind speed (corrected to hub height). See 3D turbulent wind[6.7] for details.
- **Spacing along X**: the resolution to be used for the time histories. This is expressed as the distance between successive points, which move past the turbine at the appropriate wind speed.
- **Mean wind speed**: this should correspond to the mean wind speed to be used for the simulations. In fact the turbulence characteristics are used in dimensionless form which means that they only depend weakly on wind speed. This means that the mean wind speed used here to generate the turbulence need not be the same as the wind speed used in a subsequent simulation which uses the wind field, since the turbulence will be re-scaled accordingly. However, for strict conformance to the specified turbulence characteristics, generate a different turbulent wind field, using the correct mean wind speed, for each wind speed to be used in the simulations.
- **Turbulence seed**: this is a seed for the random number generator, and should be an integer between 0 and 999. Change the seed to generate a different turbulent wind field with the same statistical characteristics.

- **Spectrum type**: select either von Karman or Kaimal, and click **Define...** to enter the detailed turbulence characteristics, as follows:

  Kaimal model: enter the lateral, vertical and longitudinal length scales to be used.

  von Karman model: the **improved model** is recommended. This conforms with ESDU data items 85020 and 86010.. Enter:

Number of components	1, 2 or 3 components of turbulence can be generated
Height above ground	The hub height is recommended
Surface roughness length	For the terrain
Latitude	Turbulence characteristics are only weakly dependent on latitude except near the equator

  The appropriate length scales and turbulence intensities are calculated and displayed, although it is possible to specify different turbulence intensities when setting up a simulation run.

  The **old von Karman model** allows either 1 or 3 components of turbulence to be generated. All the turbulence length scales must be entered by the user. This model conforms to ESDU 74031.

To start from an existing turbulent wind field definition, click **Import details…** to select an existing turbulent wind file. The parameters defining the turbulence in that file will then be loaded.

6.10.2 Advanced options

The **Advanced options** button allows the following additional functionality:

Gust iterator: this option is available only with the 'Improved von Karman' model of turbulence[6.10.1]. It will carry out an iteration in which the surface roughness is varied until a particular gust is achieved somewhere within the rotor swept area. The turbulence intensity will change with surface roughness in accordance with the model. The following parameters are required:

- **Gust wind speed, tolerance, and averaging time**: the iteration will continue until the specified maximum gust, defined as a block average with the specified averaging time, is achieved within the specified tolerance somewhere within the rotor swept area
- **Hub height and rotor diameter**: these define where the rotor swept area is in relation to the lateral and vertical extent of the wind file. Click **Get current values** to input the values currently available on the **Rotor**[4.1] screen.
- **Height of turbulent wind field**: together with the hub height, this defines the vertical position of the rotor within the wind field – see Section 6.7.
- **Minimum and maximum allowed values of surface roughness**: the iteration will fail if the required gust cannot be achieved with a surface roughness between these limits.
- **Minimum and maximum alongwind position of gust**: these can be used to ensure that the maximum gust is not too close to the start or end of the wind file.
- **Use wind shear**: If this box is checked, the wind shear will be taken into account in finding the maximum gust (which will then be most likely to occur near the top of the rotor). If used, the wind shear may be specified in terms of a shear exponent or a roughness length. Click **Get current values** to input the shear model as currently defined on the **Wind**[6.2] screen.
- **Number of files to generate**: If required, this calculation can generate more than one file with different random number seeds which match the specified criteria. In this case,

the resulting wind files will be called **name_n.wnd**, where **name** is the run name selected, and **n** is a sequential number.

6.10.3 Generating the wind field

Having defined the turbulence characteristics as above, click the appropriate button to generate the wind field either **Now** or **In Batch**. Alternatively, click the **Calculations** icon on the toolbar to bring up the Calculations screen (if it is not already present). Select **Wind turbulence generation**, and then click either **Run now,** or **Run in Batch** to store the calculation for later execution using the batch[7.2.7] facility.

6.11 Annual wind distribution

Some post-processing calculations require the annual distribution of hourly mean wind speeds to be defined. This includes the Annual Energy Yield calculation as well as any calculations which produce weighted lifetime values, for example of fatigue damage or damage equivalent loads.

Click the **Wind** icon on the toolbar and select **Annual wind distribution**. Select either a Weibull or a user-defined distribution. For the Weibull distribution, enter the mean wind speed and shape factor. A shape factor of 2 results in a Rayleigh distribution. For the user-defined distribution, enter the cumulative distribution or exceedance table, starting with an exceedance of 1.0 at zero wind speed. For wind speeds above the last point in the table, the exceedance will be assumed to decay in proportion to $\exp(-v^k)$, where v is the wind speed and k is the number entered as the “Exponent for high wind speeds”.

Click either **Cumulative plot** or **Probability plot** to view the wind speed distribution as defined.

6.12 Defining Waves

Click the **Waves** icon on the toolbar. One of the following four options must then be selected:

- **None**: no waves are required.
- **Jonswap / Pierson Moskowitz spectrum**: irregular waves with a standard energy spectrum are required.
- **User-defined spectrum**: irregular waves with a user-defined energy spectrum are required.
- **Extreme deterministic waves**: regular waves are required.

Then enter the appropriate data in the active text boxes, as described below. Any specified waves will be ignored if the turbine is not offshore - see Section 4.4.3.

6.12.1 None (no waves)

In this case the tower is not subjected to wave loading. However, if the turbine is specified as being offshore, it will still experience hydrodynamic damping and a drag force from any specified currents.

6.12.2 Jonswap / Pierson-Moskowitz spectrum

The form of the Jonswap / Pierson-Moskowitz spectrum is described in the Theory Manual. If this option is selected, enter appropriate values for the following parameters in the text boxes below the wave selection panel:

- **Peak spectral period**: the period of the most energetic component in the wave spectrum.
- **Significant wave height**: corresponding to the average height of the highest one third of the waves in the seastate. Given a time-history of wave height, this parameter can be calculated as four times the standard deviation of the water surface elevation.
- **Peakedness parameter**: this parameter controls the width of the frequency band containing most of the energy in the spectrum. It should take a value between 1 and 7. For a Pierson-Moskowitz spectrum, the peakedness parameter should be set to 1.
- **Random number seed**: an integer to be used as the seed value for the random wave generation.
- **Wave direction (from North)**: the bearing from which waves arrive at the tower. Like wind direction, wave direction is defined as the direction which the waves are coming *from*, and not the direction that the waves are going *to*. The angle increases positively to the East of North.

6.12.3 User-defined spectrum

If the wave spectrum is known at the site of interest, select this option and then click on the **Define** spectrum button. A further window opens which allows up to 100 pairs of values of frequency and power spectral density to be entered. Data entry boxes may be created and deleted by clicking on the **Add** and **Delete** buttons respectively. Data points are automatically sorted by frequency. Clicking on the **Show** button will reveal a plot of the spectrum as defined. The values of power spectral density at the lowest and highest frequencies entered should be zero.

Values for the following parameters should also be entered in the text boxes below the wave selection panel:

- **Random number seed**: an integer to be used as the seed value for the random wave generation.
- **Wave direction (from North)**: the bearing from which waves arrive at the tower.

6.12.4 Extreme deterministic waves

This option allows a regular wave train to be defined. Values for the following parameters should be entered in the text boxes below the wave selection panel:

- **Extreme wave period**: the period of the regular wave.

- **Extreme wave height**: the height of the regular wave (defined from trough to crest).
- **Wave direction (from North)**: the bearing from which the waves arrive at the tower.

6.13 Defining Currents

To define currents, click on the Waves icon on the toolbar and then select the Currents tab. Three current components may be specified, separately or in combination:

- near-surface current
- sub-surface current
- near-shore current

If more than one current is selected, the velocity vectors are added linearly. The form of each current profile is described in the Theory Manual. Any specified currents will be ignored if the turbine is not offshore (Section 4.4.3).

Click the check boxes to select the requierd current components, and enter the relevant parameters in the adjacent text boxes as described below:

6.13.1 Near-surface current

- **Mean wind speed @ 10m**.
- **Direction (from North)**: the bearing from which the current arrives at the tower. Like wind and wave directions, current direction is defined as the direction from which the current is coming *from*, and not the direction that the current is going *to*.

6.13.2 Sub-surface current

- **Surface velocity**: the observed current velocity at the sea surface.
- **Direction (from North)**: the bearing from which the current arrives at the tower.

6.13.3 Near-shore current

- **Beach slope**: the gradient of the shore, defined in degrees from the horizontal.
- **Depth at breaking waves**: the distance between mean water level and the sea bed at the location of the breaking wave.
- **Period of breaking waves**
- **Direction (from North)**: the bearing from which the current arrives at the tower. The near-shore current is assumed to flow parallel to the shore line.

7. EXECUTING WIND TURBINE CALCULATIONS

This section describes how to set up the calculation parameters which define how a particular calculation will be carried out and how to carry out the calculations themselves.

Section 7.1 describes the modal analysis calculation which is used to determine the frequencies and mode shapes of the blade and tower vibration modes. The remaining sections describe the various wind turbine performance and loading calculations themselves.

7.1 Modal analysis

A modal representation of the blade and tower structural dynamics is used in ***Bladed***. A modal analysis calculation is performed first, to compute the frequencies and mode shapes which are needed for any subsequent calculations which are to take account of blade and/or tower deflections or vibrations. The steady deflections may be of interest for calculating the Steady operational loads[7.9] or the Steady parked loads[7.10], and the vibration dynamics may be used in any of the simulations[7.4].

The modal analysis calculates uncoupled rotor and tower modes. In the subsequent calculations, the modes are coupled together by the equations of motion, so that the actual eigenfrequencies of the coupled system may change. The Model Linearisation[7.11] calculation allows the coupled modal frequencies to be calculated.

First it is necessary to decide how many modes of vibration are to be modelled.

7.1.1 Defining the modes

To define the modes required, click the **Modal** button on the toolbar. For the rotor, enter the number of out of plane and in-plane modes to be used. For the tower, enter the number of fore-aft and side-to-side modes required. Then for each mode, enter the structural damping required for each mode by clicking the **Modal damping** button.

For the rotor modes, the following additional data is required:

- **Number of pitch angles**: the modal analysis may be carried out at several pitch angles. The modal frequencies will then be interpolated for intermediate pitch angles.
- **Pitch angles**: enter the pitch angles to be used for the modal analysis.
- **Rotational speed**: supply a value representative of the rotor speed to be used in the simulations. This need not be precise, since as the rotor speed varies, the appropriate centrifugal stiffening corrections will be applied to the modal frequencies.
- **Hub rotation**: select **Locked** if the locked speed drive train[4.6.1] model is being used, or for most parked rotor calculations. Select **Free** for all other non-parked cases, and for parked cases with the dynamic drive train[4.6.2] model where there is a flexible shaft between the rotor and the shaft brake[5.11].

For the tower modes, supply also:

- **Azimuth angle** for the rotor (zero azimuth is with blade 1 pointing vertically upwards) at which the modes are to be calculated.

As the azimuth changes with rotor rotation, the modal analysis will clearly be approximate for all other positions, but in practice this inaccuracy is usually small.

It is very important to choose a self-consistent set of rotor modes, as follows:

1. For non-teetered rotors, set the number of **out of plane** modes to a multiple of the number of blades.
2. If the hub rotation is **locked** (see above), set the number of **in-plane** modes to a multiple of the number of blades. Otherwise use nB or nB-1 in-plane modes, where B is the number of blades and n is an integer.

7.1.2 Performing the calculation

Having defined the modes to be modelled as above, click the **Calculate** button. Alternatively, click the **Calculations** icon on the toolbar to bring up the Calculations screen (if it is not already present). Select **Modal analysis**, and then **Run Now**.

7.1.3 The modal frequencies

When the modal analysis is complete, click **Results** to view the modal frequencies which have been calculated. Alternatively, return to the Modal analysis screen (click the **Modal** button on the toolbar if the screen is not already present) and select **Frequencies**. These may be edited by double-clicking them if suitable field data is available. However, a more accurate approach is to modify the mass and stiffness distributions until the correct frequencies are calculated. The modal damping may be modified too at this stage.

Select a mode and click **View mode shape** for a graphical display of the modal deflections.

If some of the modes have high frequencies, it may be preferable to specify fewer modes, since high frequency modes cause the simulations to run more slowly, and tend to have less influence on loads than the modes of lower frequency.

7.2 The calculations screen

Use the **Calculations** screen to select the calculation required, to define any parameters and options required for that calculation, and to execute the calculation. If not already present, click the **Calculations** icon on the toolbar to open this screen.

7.2.1 Calculations available

The calculations are arranged in four groups:

Supporting Calculations for modal analysis[7.1] and generating turbulent wind fields[6.10].
Steady Calculations[7.3] for rapid steady-state wind turbine calculations.
Simulations[7.4] for detailed wind turbine simulations.
Post Processing[8] for further analysis of results.

First click on the desired calculation to select it.

7.2.2 Data required for calculations

Alongside each calculation is a coloured light showing whether the calculation can be performed:

Red light: No data available: no calculation possible
Yellow light: Some data available: no calculation possible
Green light: Sufficient data has been defined to perform the calculation

The box below shows all the data items required for the calculation, and indicates whether the relevant data has been defined or not. Double-clicking on any data item which is undefined opens the appropriate window so that the required data can be defined.

7.2.3 Calculation options

A number of machine and wind features, once defined, are available as options which can be switched on or off in some of the steady state calculations and simulations. Click the **Show options** button in the bottom left corner. The screen expands to show a list of the available options[7.20], with an indicator for each to show whether it can be used, as follows:

No light (grey): This option is not available for the selected calculation
Red light: No data available: option will not be used even if selected
Yellow light: Some data available: option will not be used even if selected
Green light: Sufficient data is defined for the option to be used if selected

Use the check boxes to switch on or off any desired option, provided it is available for the selected calculation. If a red or yellow dot is displayed, the option will not be used even if selected, as the necessary data has not been defined: click on the option name to display all the required data items in the box below. Double-clicking on any data item which is undefined opens the appropriate window so that the required data can be defined.

7.2.4 Specifying outputs

Some of the wind turbine calculations can potentially generate large quantities of results, and in many cases only a few of these will actually be needed. Click the **Outputs** button on the calculation screen to specify which of the available outputs[7.19] will be generated.

7.2.5 Calculation parameters

Click the Calculation parameters[7.5] button on the calculation screen to change or define the specific parameters required to define the selected calculation. The following additional data may also be defined:

- Physical constants[7.16] : to define air density and viscosity, and gravitational acceleration.
- Aerodynamics control[7.15] : to define the aerodynamic modelling parameters.
- Simulation control[7.13] : to define how a simulation will proceed.
- Safety factors[7.17] : to be applied to certain loads.
- Imbalances and pitch failures[7.18]: to specify mass and aerodynamic imbalances and pitch system failures.

7.2.6 Executing a calculation

Click on **Run Now** to carry out the selected calculation immediately, or **Run in Batch** to store the calculation for later execution by the batch facility. Except for certain calculations, a window appears in which a directory can be selected into which the calculation outputs will be placed. If the directory does not exist, it will be created. A run name is also required. Thus a number of calculations may be placed in the same directory, each with a different run name.

There is also the option to select ASCII or binary format for the calculation results. Binary format gives much smaller files, and should be faster and more accurate. Graph plotting, post-processing and tabulation all work with either format. If you want to see the raw numbers from binary files, you can still tabulate[9.9] the numbers (to the screen, an ascii file, Excel or Word).

If a whole series of calculations is to be performed, for example a set of simulations for different load cases which might include stops and starts as well as normal running and parked conditions in a series of different wind conditions, some thought should be given to the directory structure used, and the file extensions used within each directory. For example a tree structure could be used with a sub-directory for each group of load cases. Each individual load case could then be in its own sub-directory, or different extensions could be used within the directory, for example to indicate different mean wind speeds. Bear in mind also the need for post-processing results to be stored for each load case. Some post-processing results may also be summed over a series of load cases.

Calculations may also be set off from the **Calculations** pull-down menu on the toolbar. Post-processing and supporting calculations may also be set off from the relevant data entry screen.

7.2.7 Batch processing

Some calculations, notably long simulations with high frequency dynamics, may take as long as several hours to run. Use the batch system to store the complete details of one or more calculations which can then be run at a later time, for example overnight. Use the **Run in Batch** button instead of the **Run now** button on the **Calculations** window. When all the required calculations have been added to the batch, the calculations may be started using the **Batch List** window, accessed from the **Batch** menu item on the main toolbar.

A number of different batch queues may be maintained. Each batch must be in a separate directory.

The **Batch** menu item on the main toolbar gives access to the **Batch List** screen. This shows all the calculations in the batch queue, and allows the user considerable flexibility in managing batch queues, as described below.

- Directly from the batch list, you may: View the status of each batch job.
- Enable or disable any batch job by double-clicking on it.
- Select one or more jobs for subsequent operations.
- Delete one or more jobs from the batch list.
- Move a job up or down the queue using the arrow buttons.
- Start the batch run.
- Abort the entire batch run.
- Clear the Batch List.

From the **Batch** pull-down menu, you may:
- Change to another batch queue in a different directory.
- Start the batch run.
- Abort the entire batch run.
- Clear the Batch List.

From the **Jobs** pull-down menu, you may:
- Select all jobs in the queue.
- Enable, disable or reset selected jobs.
- Delete selected jobs.
- Run the selected job.
- Abort the job currently running.
- View any warning or error messages from completed jobs.
- View the results of completed jobs.

From the **Edit** pull-down menu, you may:
- Load the calculation details for a selected job. If this is a nested batch (see below), the nested batch will be opened in the batch window.
- Import particular details from a project file or calculation details file into the currently selected job(s). You will be prompted for the file to import from, and asked whether to overwrite the selected batch jobs or create new ones.
- Resubmit the selected job (for example after loading the calculation details and editing them).
- Redirect the output of the currently selected job(s). If more than one job is selected, you will have the option of respecifying the output individually, or simply maintaining the entire directory structure and run names but specifying a new root directory.
- Create batch jobs from existing calculation details. You may choose to import a single calculation into the batch queue, or all calculations in a given directory, together with subdirectories if desired.
- Copy the selected job(s) into a buffer for subsequent pasting into another batch queue.
- Paste jobs from the buffer into the current batch.
- Move a selection of jobs to a different place in the queue.

- Add a nested batch, i.e. another complete batch to be executed at this point in the current batch. This allows a 'batch of batches' to be specified. A nested batch may itself contain further nested batches, with no limit to the number of levels of nesting.

From the **Options** pull-down menu, you may:

- Select the **Start batch jobs in foreground** option. With some operating systems, this may considerably increase the speed of calculations.

7.2.8 Retrieving calculation details

From the **File** menu on the main toolbar, select **Open Project. Use the File Type** selector to request Calculation Details files, which have a $pj extension and reside in the output directory for the run. The calculation details will be loaded and can be inspected. This is useful if a run is to be repeated exactly or with modifications. If desired, the calculation details may be saved as a new project file.

Calculation details may also be retrieved directly from jobs in the Batch List[7.2.7].

7.3 Steady calculations

These calculations give quick results by ignoring the effects of any turbine dynamics and wind turbulence. They are thus particularly useful at the preliminary design stage.

- Aerodynamic information[7.6] : This gives the aerodynamic parameters including local aerodynamic loading at each blade station, assuming uniform steady wind flow.
- Performance coefficients[7.7] : Power, torque and thrust coefficients are calculated assuming a uniform steady wind field.
- Steady power curve[7.8] : This gives power, torque and thrust as a function of wind speed, assuming a uniform steady wind field.
- Steady operational loads[7.9] : This calculates all the loads in a uniform steady wind field.
- Steady parked loads[7.10] : Loads on the parked turbine are calculated in steady wind. As the blade position is specified, the wind field need not be spatially uniform.
- Model linearisation[7.11]: This calculation is only available with an additional license. Small perturbations are used to generate system responses, from which the Linear Model[8.19] post-processing calculation is then able to derive a linearised model of the turbine in state-space form. This is of particular value for designing controllers.

7.4 Simulations

Simulations are used to calculate the loads, deflections and other parameters of the wind turbine as a function of time, using the full dynamics of the system, driven if desired by a time-varying wind[6.1] input.

They may be used, for example, to simulate the behaviour of a turbine for period of, say, 10 minutes at each of a number of wind speeds, so that the resulting loads can be analysed to produce a representative fatigue load spectrum for the turbine.

The following simulations may be run:

- **Power production** (i.e. normal running)
- **Normal stop**
- **Emergency stop**
- **Start-up**
- **Idling**
- **Parked**

The **Dynamic power curve**[7.14] calculation is also available. This is a sequence of Power Production simulations at different wind speeds.

The full dynamics of the turbine will be used for all these simulations, together with the appropriate supervisory control[5] parameters which define the stopping and starting logic, and the state of the machine when idling or parked. See also Yaw manoeuvre[5.17.2] to cause the turbine to yaw during a simulation.

To define the length of the simulation, and other parameters which control how the simulation proceeds, see Simulation control[7.13].

7.5 Calculation parameters

Click the **Calculation parameters** button on the **Calculations** screen to define the parameters for controlling particular calculations. These include **general** parameters as well as parameters for defining **specific** calculations.

General parameters:

- Physical constants[7.16] : to define air density and viscosity, and gravitational acceleration.
- Aerodynamics control[7.15] : to define the aerodynamic modelling parameters.
- Simulation control[7.13] : to define how a simulation will proceed.
- Safety factors[7.17] : to be applied to certain loads.
- Imbalances and pitch failures[7.18] : to specify mass and aerodynamic imbalances and pitch system failures.

Parameters for specific calculations:

- Aerodynamic information[7.6].
- Performance coefficients[7.7].
- Power curve[7.8].
- Steady operational loads[7.9].
- Steady parked loads[7.10].
- Model linearisation[7.11].

7.6 Aerodynamic information calculation

This is a simple steady-state calculation which outputs the following aerodynamic information at each blade station:

- **Inflow factors**: axial and tangential
- **Inflow angle**
- **Angle of attack**
- **Tip loss factor**
- **Reynolds number**
- **Lift, drag and pitch moment coefficient**
- **Relative wind speed**
- **Aerodynamic loadings:** in plane force, out of plane force, and pitching moment per unit length of blade.

Use the **Aerodynamic information** screen (see Calculation parameters[7.5]) to define the following parameters for the calculation:

- **Wind speed**
- **Pitch angle**
- **Rotational speed** of the rotor.

7.7 Performance coefficients calculation

This calculation generates dimensionless power, torque and thrust coefficients for the rotor as a function of tip speed ratio, assuming a uniform steady wind.

Use the **Performance coefficients** screen (see Calculation parameters[7.5]) to define the following parameters for the calculation:

- **Minimum tip speed ratio**
- **Maximum tip speed ratio**
- **Tip speed ratio step**: the tip speed ratio interval required
- **Pitch angle**
- **Rotational speed** of the rotor.

The wind speed will be varied to give the required tip speed ratios at the specified rotational speed. Although the performance coefficients are dimensionless, they may depend on the wind speed or rotational speed, and not just on the tip speed ratio, because of Reynolds number effects.

7.8 Power curve calculation

Both steady-state and dynamic calculations can be used to generate a power curve. The **steady-state power curve** calculation generates the wind turbine power curve as a function of wind speed, assuming uniform steady winds. Shaft and electrical power, gearbox torque and thrust are calculated. The **dynamic** power curve calculation performs a simulation at each wind speed to produce a mean power curve with all dynamic effects (including controller dynamics and the effects of turbulence) included.

Use the **Power curve** screen (see Calculation parameters[7.5]) to define the following parameters for the calculation:

- **Minimum wind speed**: the low wind cut-in speed
- **Maximum wind speed** the high wind cut-out speed
- **Wind speed step**: the wind speed interval required

The remaining parameters are relevant for the **steady-state power curve** calculation only:

- **Calculate pitch and speed change**: select **Yes** to use the schedule[7.12] defined by the generator[4.8] and controller[5] to define the pitch angle and rotor speed at each wind speed (see Section 6.21). Select **No** to calculate a simple power curve at fixed pitch angle and rotor speed.

If **No** is selected, also supply the following:

- **Fixed pitch angle**
- **Fixed rotational speed** of the rotor.

7.9 Steady operational loads calculation

This calculation generates wind turbine loads as a function of wind speed, assuming uniform steady winds. This optionally includes blade, tower and other loads - see Output Control[7.19]. It also includes the calculation of partial derives which are useful for control design, and in particular for gain scheduling[5.6].

Use the **Steady loads** screen (see Calculation parameters[7.5]) to define the following parameters for the calculation:

- **Minimum wind speed**: the low wind cut-in speed
- **Maximum wind speed** the high wind cut-out speed
- **Wind speed step**: the wind speed interval required
- **Calculate pitch and speed change**: select **Yes** to use the schedule[7.12] defined by the generator[4.8] and controller[5] to define the pitch angle and rotor speed at each wind speed (see Section 6.21). Select **No** to calculate a simple power curve at fixed pitch angle and rotor speed:

If **No** is selected, also supply the following:

- **Fixed pitch angle**
- **Fixed rotational speed** of the rotor.

7.10 Steady parked loads calculation

This calculation generates wind turbine loads for a parked rotor in steady winds. This optionally includes blade, tower and other loads - see Output Control[7.19].

Use the **Steady parked loads** screen (see Calculation parameters[7.5]) to define the following parameters for the calculation:

- **Steady wind speed**: note that although the wind speed is steady, it need not be spatially uniform - see Defining the wind[6.1].
- **Azimuth angle**: the rotor azimuth position; zero means that blade 1 is pointing vertically upwards.
- **Yaw angle**: the nacelle angle measured clockwise from North, assuming that the wind is blowing from the North: see wind direction[6.9].
- **Wind inclination**: for non-horizontal wind flows (e.g. on the side of a hill). A positive value indicates a rising wind.
- **Pitch angle**: the pitch angle or aileron/flap/airbrake deployment angle - see Rotor[4.1].

The calculation also allows a sweep through any one of the four above angles, i.e. azimuth, yaw, wind inclination or pitch angle. Use **Parameter to vary** to define which angle will vary. The sweep start at the value specified above for that parameter. Specify **End value** to define the end of the sweep, and **Step** to define the step size.

7.11 Model linearisation calculation

This calculation has two alternative forms:

- Campbell diagram
- Model Linearisation

Use the **Model linearisation** screen (see Calculation parameters[7.5]) to select the calculation type and to define the parameters required for these calculations.

Campbell diagram
The Campbell Diagram calculation is available for all users with a Simulation licence or an Educational licence. It generates state perturbations, and uses the resulting variations in the state derivatives to compute the eigenfrequencies of the coupled system. This is repeated for a series of operatring points, and the eigenfrequencies are then displayed on a graph as a function of rotor speed.

The operating points are defined in two ranges. For low rpm, the turbine is assumed to be idling. Operating points are defined at a range of rotor speeds, at a specified wind speed. The idling pitch angle is defined by the Idling Conditions[5.15]. For power production, a range of wind speeds operating points is defined, and the rotor speed and pitch angle will then be determined by the power train and control system definition[7.12], in the same way as for the Steady Power Curve[7.8]. A reasonable number of rotor speeds and wind speeds should be defined, to obtain a useful number of points on the Campbell diagram.

For each wind speed and rotor speed, it is also possible to specify a range of azimuth angles, but this is not normally useful since the system dynamics are usually not significantly dependent on azimuth.

The parameters required are as follows:

- **Rotor speed operating points**: define the minimum and maximum rotor speeds required in the idling range, and the rotor speed interval.
- **Wind speed operating points**: define the minimum and maximum steady wind speeds required in the power production range, and the wind speed interval.
- **Azimuth operating points**: in most cases, only one azimuth angle need be used.
- **Idling Wind Speed**: the wind speed to be used for the idling range.
- The **Maximum frequency** to be plotted on the Campbell diagram.
- The **Minimum correlation coefficient**. If the correlation between a particular state and state derivative is below this value, a zero relationship is assumed between them. The recommended value is 0.8.

Model Linearisation

The Model Linearisation calculation is only available for users with a licence for the Linearisation module. It generates input and state perturbations, and records the resulting variations in the state derivatives and the selected outputs. This is done for a series of steady-state power production operating points. The Linear Model[8.19] post-processing calculation is then able to derive a linearised model of the turbine in state-space form. This is of particular value for designing controllers.

The parameters required are as follows:

- **Wind speed operating points**: define the minimum and maximum steady wind speeds required, and the wind speed interval.
- **Azimuth operating points**: in most cases, at least for 3-bladed rotors, the rotor azimuth is unimportant, and only one azimuth angle need be used.
- **Wind speed perturbation**
- **Pitch perturbation**
- **Generator torque perturbation**

Click **Set default perturbations** to obtain sensible default values. It is unlikely that these will need to be changed, unless the resulting model indicates problems of poor resolution (perturbations may be too small) or hitting non-linearities (perturbations may be too large).

If required, it is possible to perturb the wind speed and pitch separately for each blade.

7.11.1 Selection of model features for Model Linearisation and Campbell Diagram

It is important to consider carefully the features of the turbine model which should be included in the model linearisation calculation. A few examples are given below:

- Clearly if the model is to be used to design a control algorithm, it must be an open-loop model. For example, if designing a pitch controller, the built-in pitch PI controller[5.2.2,5.4.2] should not be used, or the gain should be set to zero. Any external controller will not contribute any dynamics as it operates in discrete time, so it will effectively be open-loop.
- Azimuth-dependent and certain other features may not be desired. For example
  - flow inclination, wind shear[6.2], tower shadow[6.3] and upwind turbine wake[6.4]
  - yaw motion (especially with friction)[5.17]
  - gravity loads, safety factors[7.17]
  - Imbalances and pitch failures[7.18]

  Most of these may be switched off using the Calculation Options[7.20]

- Highly non-linear features may prevent a suitable linear model from being generated. Examples might include tight pitch rate limits, generator torque limits, and the more complex pitch actuator models[5.10], especially if they involve a ramp control, bearing friction, torque limits, etc. In such cases it is advisable to substitute a linear approximation to the actuator dynamics, for example a second-order passive model tuned to give a similar step response. (Step responses may be generated using a simple external controller to generate step changes in demand.)
- For the model linearisation calculation, it is usually desirable to find steady-state values for all the states at each operating point. To do this, switch on the **Refine Initial States** option in **Simulation control**[7.13].

7.12 Pitch and speed schedule

For Steady Power Curve and Steady Operating Load calculations, the pitch and rotor speed schedule represents the appropriate steady-state pitch (or aileron) angle and rotor speed corresponding to any particular wind speed. The schedule is defined by:

- the steady-state Controller[5] for the pitch angle, and also for the rotor speed in the case of a variable speed turbine, and
- the Generator[4.8] for the rotor speed in the case of a fixed speed turbine.

7.13 Simulation control

Use the **Simulation control** screen (see Calculation parameters[7.5]) to define various parameters which control the progress of a simulation. These parameters are defined below:

- **Output time step**: this is the time interval at which the output variables are sampled. For individual simulations these samples are stored in the output files. For the dynamic power curve[7.14] calculation, these samples are used to calculate the statistics for each individual simulation (i.e. each wind speed).
- **Time to start writing output**: Some simulations may start with a small transient, if the initial conditions are not calculated to sufficient accuracy. In this case it may be desirable to ignore the first part of the simulation until the transient has settled. This parameter specifies the length of time to ignore at the start of the simulation before output samples start to be taken.
- **Simulation end time**: This parameter gives the maximum time for which the individual simulation will run. Normal and emergency stop simulations may end earlier if the rotor has stopped and any extra time has elapsed.
- **Time to begin a stop**: for normal and emergency stop simulations, this is the length of time for which normal running continues before the built-in stop logic is initiated.
- **Extra simulation time after stopping**: for normal and emergency stop simulations, this is the length of time for which the simulation continues after the rotor has come to rest.
- **Start time for turbulent wind**: this applies to simulations which use a turbulent wind[6.7] input. It is the time into the turbulence time history at which the simulation begins. Specify zero to start at the beginning of the turbulence history.
- **Blade station economiser**: this can be used to speed up simulations, by setting a value greater than zero. Then the full aerodynamic calculations will only be carried out at a few

blade stations on trial integrator steps, while other stations will be interpolated. All stations are fully calculated at each complete integrator step, so there is very little loss of accuracy. Enter a percentage of the rotor radius representing the approximate interval between stations at which full calculations will always be carried out. A value of about 35% is recommended. Set zero to ensure maximum accuracy.

- **Refine initial states**: This option carries out an initial iteration to find the steady-state values of the states. This may help to prevent transients at the start of a simulation, and is also particularly recommended for the **Model linearisation**[7.11] calculation. The iteration does not include the yaw state at present.

The following parameters may be used if necessary to fine-tune the performance of the variable-step integrator:

- **Integrator tolerance**: this parameter defines the accuracy of the simulation. A value of 0.005 is recommended to start with, but in special cases this may need to be reduced to improve the accuracy, at the expense of slower simulations. The usual symptoms of insufficient accuracy are unexplained spikes on some variables which are not reflected in related variables. If there is any doubt, repeat a simulation with a different tolerance to see if the results are significantly affected.
- **Minimum time step**: the simulation uses a 4/5th order Runge-Kutta variable time step method. The time step will be reduced automatically if the specified tolerance is exceeded, until this minimum value is reached. A very small value for the minimum time step is recommended, such as 10^{-8} s, to ensure that the accuracy of simulation is not constrained by this. In special cases, increasing the minimum time step may speed up the simulation with little loss of accuracy, but it is advisable to check that the results are not significantly altered by doing this. This situation may arise for example with a dynamic mode which is inactive because it is heavily damped. It may be better to remove the mode completely.
- **Maximum time step**: This should normally be the same as the **Output time step**, although a smaller value might be useful in some cases if the output time step is particularly long.
- **Initial time step**: The recommended value is zero: the minimum time step will in fact be used. A value larger than the minimum time step will speed up the initialisation of the simulation, but there is a risk of numerical problems if too large a value is used.

7.14 Dynamic power curve calculation

The **Dynamic power curve** calculation carries out a series of simulations at different wind speeds to generate a power curve which takes into account all the dynamics of the system as well as the effects of turbulence. To define the wind speeds to be used, see Power curve calculation[7.8].

For better control of wind conditions, it is recommended that instead of using **Dynamic power curve** calculation, a series of separate power production simulations is run for a number of wind speeds. The **Annual energy yield**[8.13] calculation then constructs the dynamic power curve from these simulation results, and simultaneously computes the annual energy yield.

The normal simulation outputs are not generated for each individual simulation. Instead, the output samples for certain variables which would normally be saved to output files are used instead to calculate the mean, maximum, minimum and standard deviation of that quantity at each mean wind speed, and it is these statistics which are saved as output. As well as shaft and electrical power, the statistics of gearbox torque, thrust, pitch angle and hub wind speed are also generated. To define the sampling frequency and length of each individual simulation, see Simulation control[7.13].

Caution: this calculation uses a single turbulent wind history, scaled for each wind speed. Since the simulation time is constant, the high wind speed simulations will use more of the turbulence history than the low wind speed simulations. When only a part of the wind speed time history is used, the mean wind speed obtained may be different from the nominal wind speed for that simulation. Therefore it is important to plot the resulting power curve against the actual mean wind speed obtained, and not against the nominal wind speed. Note also that the actual mean wind peed obtained is the mean of the magnitude of the wind vector, taking into account all three components of turbulence (if used), which is greater than either the mean longitudinal component, or the mean horizontal component as measured by a cup anemometer.

7.15 Aerodynamic models

Use the **Aerodynamics control** screen (see Calculation parameters[7.5]) to define various parameters which define how the aerodynamic calculations are carried out.

Parameters applicable to all calculations:

- **Prandtl corrections**: Select whether or not to apply a Prandtl correction for **tip loss** and **hub loss** effects.
- **Allowable error in flow induction factors**: specify the accuracy required for the induction factors. A value of 0.001 is usually suitable. Smaller values result in slower calculations but greater accuracy.

The inflow calculation is inappropriate for parked and slow idling cases, and may be switched off at low tip speed ratios. For
Zero inflow below tip speed ratio enter the tip speed ratio below which the inflow calculation is switched off; for
Full inflow above tip speed ratio enter a tip speed ratio above which the full inflow calculation will be carried out. Between these two values the inflow calculation is phased in gradually.

Parameters applicable to dynamic calculations only:

- **Wake models:**
  The **equilibrium** wake model recalculates equilibrium inflow velocities at every instant in time.
  The **frozen wake** model calculates equilibrium inflow velocities at the start of the simulation, but then maintains these values for the rest of the simulation. This results in a fast calculation, and may be sufficiently accurate for simulations at near-zero yaw angles with a relatively uniform wind field and little variation in wind speed or rotor speed.
  The **dynamic inflow** model recalculates the inflow factors at intervals, and uses an induction lag. This model is recommended for simulations in which the wake cannot considered to be frozen or unchanging, and it results in faster calculations than the equilibrium wake model. **Dynamic wake time step**: specify the time interval at which the inflow is recalculated. To avoid slowing simulations unnecessarily, it is recommended to

use a time step equal to the simulation output timestep, or the external controller timestep. As the wake only changes slowly, the precise value selected is not critical.

- **Stall hysteresis**
  The check box allows the Beddoes stall hysteresis model to be selected, which allows dynamic stall effects to be modelled. Enter the:
- **Separation position time constant**: This is a dimensionless time constant, given in terms of the time taken to travel half a chord. It defines the lag in the movement of the separation point due to unsteady pressure and boundary layer response. It should have a negative value, and -3.0 is recommended.
- **Starting radius**: This is the radial position, given as a percentage, outboard of which the dynamic stall model will be used. This allows the use of data on the inboard blade sections that has been corrected for the effects of three dimensional flow. This type of data should not be used in combination with the dynamic stall model which has been developed for use with two dimensional aerofoil data.

7.16 Physical constants

Use the **Physical constants** screen (see Calculation parameters[7.5]) to define the following general physical constants:

- **Air density**
- **Air viscosity** (dynamic)
- **Gravitational acceleration (g)**
- **Density of water (only required for offshore turbines)**

7.17 Safety factors

Use the **Safety factors** screen (see Calculation parameters[7.5]) to define additional factors to be applied to the blade, hub and tower loads when they are output. Each load may have several components, and different safety factors may be applied to each, as follows:

- **Aerodynamic loads**: this safety factor will multiply the aerodynamic component of output loads.
- **Gravity loads**: this safety factor will multiply the gravitational component of output loads.
- **Inertial loads**: this safety factor will multiply the inertial component of output loads.
- **Wave loads**: this safety factor will multiply the hydrodynamic component of output loads (for offshore turbines).

Note: the use of these partial safety factors is no longer recommended. Safety factors should rather be applied during post-processing of the loads. No safety factors are applied to other outputs, such as drive train, generator and summary information, deflections, velocities, etc.

7.18 Imbalances and Pitch Failures

Use the **Imbalances and Pitch Failures** window (see Calculation parameters[7.5]) to define any mass imbalance of the rotor, or errors in the set angle or the pitch angle of one or more blades, and to specify pitch failure cases. Enter the following data:

- **Imbalance mass**: the part of the rotor mass which is out of balance. **Note:** this is not an *additional* mass: the total rotor mass is not changed by this parameter.
- **Radius of imbalance**: the radial position of the centre of gravity of the imbalance mass.
- **Azimuthal position of imbalance**: zero means the imbalance mass centre of gravity is on blade 1.
- **Time for pitch failure**: if **Pitch failure mode** (see below) is set to '**At time T**', enter the time at which the failure will occur.
- **Error in blade set angle**: for each blade, specify the error in set angle - see Rotor[4.1].
- **Error in pitch angle**: for each blade, specify the error in pitch (or aileron/airbrake) angle - see Blade geometry[3.2].
- **Error in blade azimuth**: for each blade, specify the error in azimuthal position for each blade. (Note: this will result in an additional contribution to the mass imbalance.)
- **Pitch failure mode**: for each blade, specify whether the pitch mechanism has **Failed**, is operating correctly (**Moving**), or is set to fail at the **Time for pitch failure** specified above. In the latter case, the blade pitch will freeze at its current value at that time.
- **Pitch of failed blade**: for each blade where the pitch mechanism is **Failed**, specify the angle at which the pitch is stuck (applies also to ailerons/airbrakes).

7.19 Controlling the calculation outputs

A large amount of output is potentially available from some of the steady state calculations and simulations. Click the **Outputs** button on the **Calculations** screen to define which of these outputs are required. The outputs are grouped into **blade, tower** and **other** outputs.

Aerodynamic information, Performance coefficients and the Power curve calculations are unaffected, as they produce a pre-defined set of outputs.

7.19.1 Blade outputs

Click **Blade outputs** on the output specification screen.

Blade outputs include **aerodynamic information** (including distributed aerodynamic loadings), **blade loads** and **deflections**. For each of these categories, the information may be generated at any or all of the blade stations[3.1]. For each blade station, click the box to indicate whether or not that information is required.

Having defined the blade stations required, specify for each individual output whether that information is required on the **First blade, All blades**, or not at all (**None**). For the loads, this can be specified independently for each of the following loads:

- **Bending moments** and **shear forces**, resolved in out of plane and in-plane directions. These loads can also be requested resolved into flapwise and edgewise directions, which are in the local blade axis system, i.e. resolved perpendicular and parallel to the local blade chord.

- **Radial forces**.
- **Pitch moments**.

For the blade deflections, this can be specified independently for in-plane and out of plane deflections.

7.19.2 Tower outputs

Click **Tower outputs** on the output specification screen.

Tower loads may be generated at any or all of the tower stations[4.4]. For each tower station, click the box to indicate whether or not the loads are to be output there.

Having defined the tower stations required, specify for each individual output whether that information is required. For the loads, this can be specified independently for each of the forces and moments.

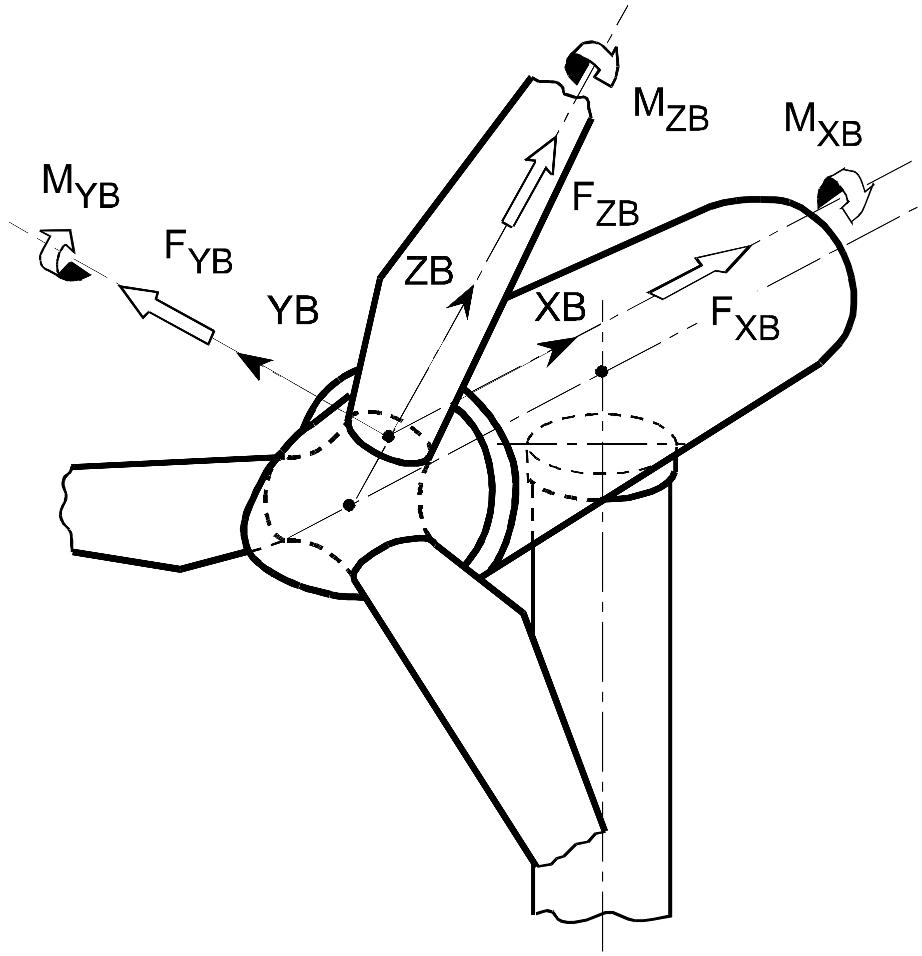
7.19.3 Other outputs

Click **Other outputs** on the output specification screen, and select which of the output files are required. These include, for example,

- **Hub loads** (rotating with the rotor)
- **Hub loads** (fixed frame of reference)
- **Drive train information**
- **Generator information**
- **Controller information**
- **Summary information**: includes teeter, yaw and pitch angles, rotor azimuth, aerodynamic, gearbox and brake applied torque, hub wind speed, rotor speed, power output.

7.19.4 Co-ordinate systems

The co-ordinate system for the loads and deflections is defined in Figures 6.28.1 to 6.28.3 on the following pages.

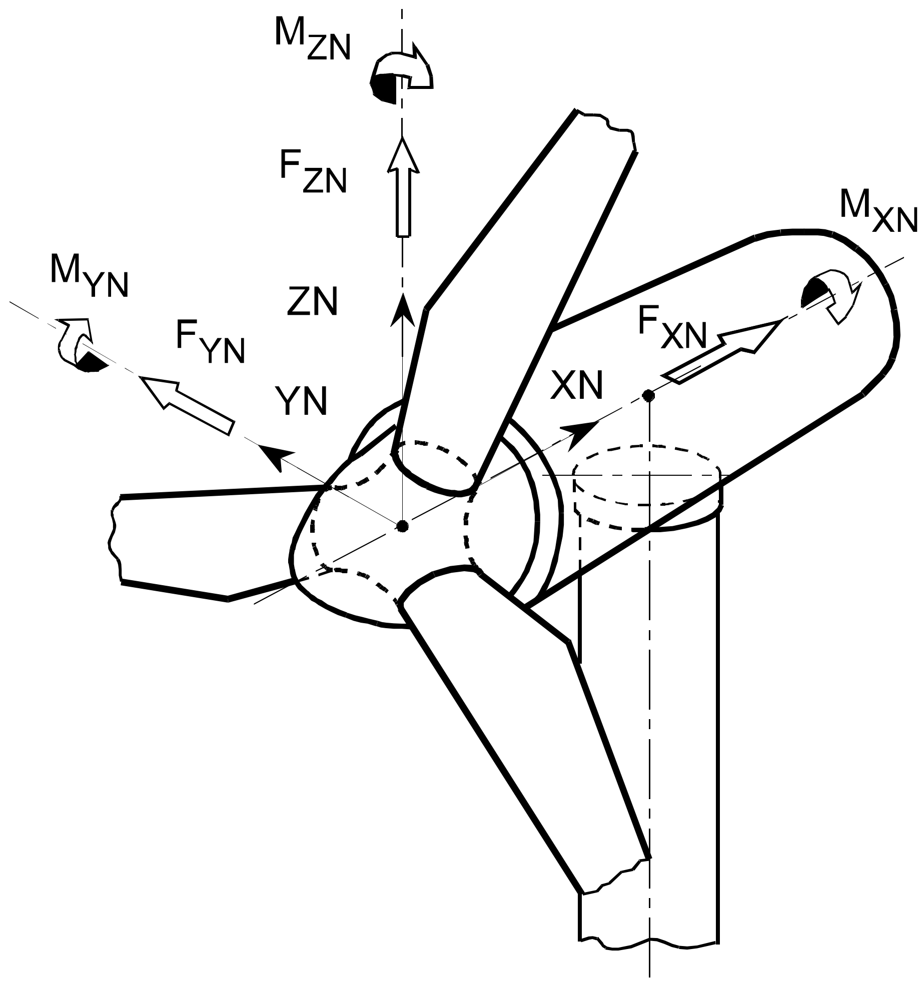


ZB Radially along blade pitch axis.
XB Perpendicular to ZB, and pointing towards the tower for an upwind turbine, or away from the tower for a downwind turbine (the picture shows an upwind turbine).
YB Perpendicular to blade axis and shaft axis, to give a right-handed co-ordinate system independent of direction of rotation and rotor location upwind or downwind of the tower.

Origin At each blade station.

Figure 6.28.1: Co-ordinate system for blade loads and deflections

Blade loads may also be output in local co-ordinates, where X is perpendicular to the local chord line, Y is parallel to the chord, and Z is along the blade pitch axis. The local chord line changes with set angle, twist and pitch angle.



Hub loads in fixed frame of reference:

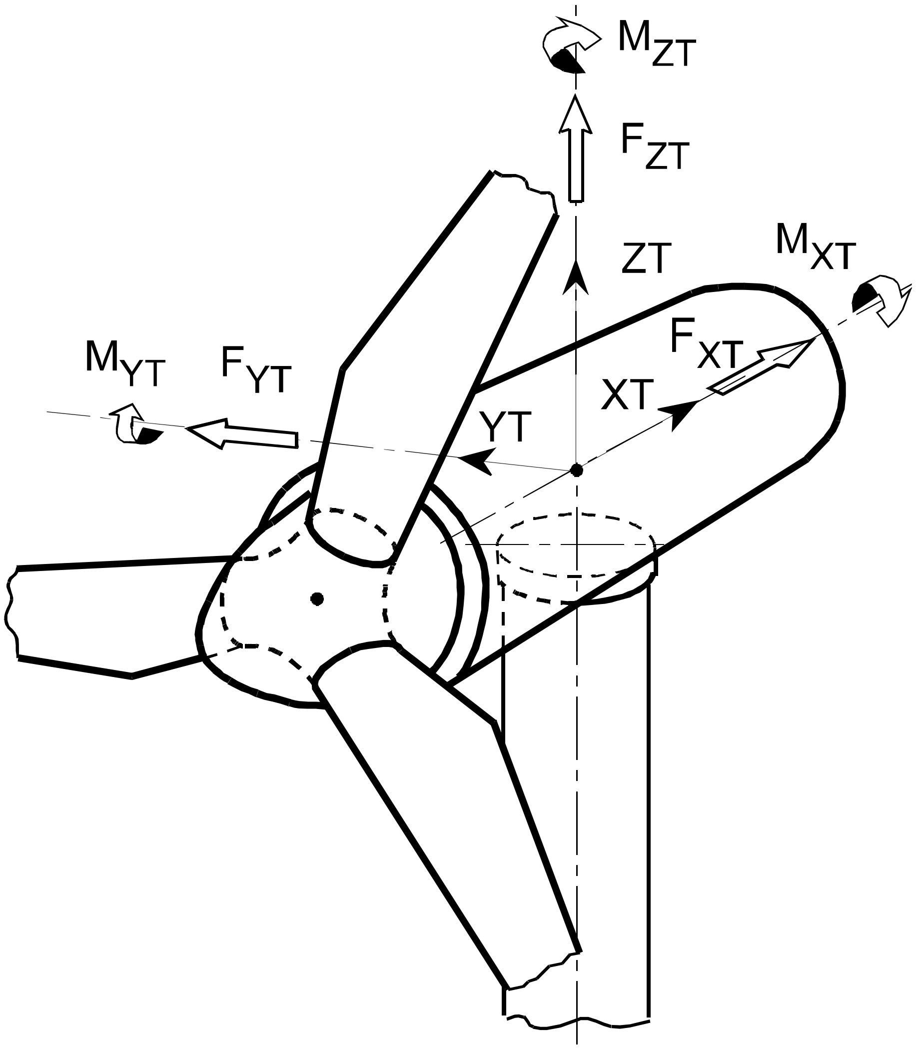
XN Along shaft axis, and pointing towards the tower for an upwind turbine, or away from the tower for a downwind turbine (the picture shows an upwind turbine).
ZN Perpendicular to XN, such that ZN would be vertically upwards if the tilt angle were zero.
YN Horizontal, to give a right-handed co-ordinate system independent of direction of rotation and rotor location upwind or downwind of the tower.

Hub loads in rotating frame of reference:

XN Along shaft axis, and pointing towards the tower for an upwind turbine, or away from the tower for a downwind turbine (the picture shows an upwind turbine).
ZN Perpendicular to XN, such that ZN would be aligned with blade 1 axis if the cone angle were zero.
YN Perpendicular to XN and ZN, to give a right-handed co-ordinate system independent of direction of rotation and rotor location upwind or downwind of the tower.

Origin At hub centre (intersection of blade and shaft axes).

Figure 6.28.2: Co-ordinate system for hub loads



XT	Pointing South.
ZT	Vertically upwards.
YT	Pointing East.

Origin At each tower station.

Note that for steady-state calculations the wind is deemed to blow from the North.

Figure 6.28.3: Co-ordinate system for tower loads and deflections

The yaw bearing is assumed to be located at the top tower station. The co-ordinate system for yaw bearing loads is the same as for the top tower station except that it rotates with the nacelle. The co-ordinate system for tower top motion is the same as for the yaw bearing.

7.20 Specifying calculation options

A number of machine and wind features, once defined, are available as options which can be switched on or off in some of the steady state calculations and simulations. Click the **Show Options** button in the bottom left corner of the **Calculations** screen to display the options.

Initially, all options are switched on by default, but if any are switched off for a given calculation, the setting will be remembered. An option displaying a yellow or red light will not be used even if selected – see Section 7.2.3 . The options available are:

- **Energy losses**[4.9]**.**
- **Drive train flexibility**[4.6.2]**.**
- **Drive train mounting flexibility**[4.7]**.**
- **Teeter restraint**[4.3]**.**
- **Teeter motion**: see hub[4.2]**.**
- **Blade modes** (see vibrational modes)[7.1].
- **Tower modes** (see vibrational modes)[7.1].
- **Passive yaw motion**[5.17].
- **Prescribed yaw manoeuvre**[5.17.2].
- **Imbalances and pitch failures**[7.18].
- **Load safety factors**[7.17].
- **Gravity loads**. See also safety factors[7.17].
- **Inertial loads**. See also safety factors[7.17].
- **Nacelle and tower windage loads**.
- **Tower shadow**[6.3]**.**
- **Wind shear**[6.2]**.**
- **Upwind turbine wake**[6.4]**.**

8. POST-PROCESSING

Bladed provides an integral post-processing facility for analysis of calculation results. The following calculations are provided:

- Basic statistics[8.1] to calculate the mean, minimum, maximum, standard deviation, skewness and kurtosis of a signal.
- Fourier harmonics[8.2] to calculate the fourier components of a signal at multiples of rotor rotational frequency.
- Periodic components[8.3] to separate out the periodic and stochastic parts of a signal.
- Extreme predictions[8.4] to estimate lifetime extreme loads from a sample time history.
- Auto spectrum[8.5] to calculate the auto-spectral density (frequency spectrum) of a signal.
- Cross spectrum[8.6] to calculate cross-spectral density, coherence and transfer functions between any two signals.
- Probability density[8.7] to calculate the probability distribution of a signal.
- Peak analysis[8.8] to calculate the probability density of signal peaks and troughs.
- Level crossing analysis[8.9] to calculate the probability of crossing any particular threshold.
- Rainflow cycle count[8.10] for cycle counting of a signal for fatigue analysis, and calculation of damage equivalent loads.
- Fatigue analysis[8.11] to calculate fatigue damage.
- Annual energy yield[8.13] to calculate the annual energy yield as a function of mean wind speed from a power curve.
- Channel combination[8.14] to combine and scale a number of signals (useful to produce a combined stress signal for fatigue analysis).
- Ultimate loads[8.16] to find the maximum and minimum values of loads and concurrent values of other loads.
- Ultimate load cases[8.17] to identify the load cases producing maximum and minimum values of specified loads.
- Flicker[8.18] to calculate the flicker severity due to voltage variations on the network caused by the wind turbine.
- Linear model[8.19] to convert the output of the Model Linearisation[7.11] calculation into a state-space model suitable for control design, for example using Matlab [2].

Post-processing calculations may be run from the **Calculation** screen, by selecting the calculation required and pressing **Run now** or **Run in Batch**, or from the **Post-processing** screen (obtained by clicking the **Analyse** icon on the toolbar) by pressing the **Execute now** or **Add to Batch** buttons.

8.1 Basic statistics

This calculation generates the mean, minimum, maximum, standard deviation, skewness and kurtosis of a signal.

Click the **Select...**[8.20] button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case.

Note that if the skewness and kurtosis are not required, the remaining statistics may be obtained directly from the **Data View**[9] facility

8.2 Fourier harmonics

This calculation generates the fourier components of a signal at multiples of rotor rotational frequency.

Click the **Select...**[8.20] buttons to define the signal to be processed and the signal representing rotor azimuth. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case. Then choose either automatic or manual selection of parameters. Automatic selection is usually adequate, but if entered manually, the parameters required are:

- **Number of bins**: the number of azimuthal bins to be used in the calculation (minimum 4, maximum 144). More bins will generally give a more accurate result, but do not choose more bins than the number of signal samples in one revolution of the rotor.
- **Number of harmonics**: the number of harmonics to be calculated (minimum 1, maximum one quarter of the number of bins).

8.3 Periodic components

This calculation separates out the periodic and stochastic parts of a signal.

Click the **Select...**[8.20] buttons to define the signal to be processed and the signal representing rotor azimuth. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case. Then choose either automatic or manual selection of data. Automatic selection is usually adequate, but if entered manually, the parameter required is:

- **Number of bins**: the number of azimuthal bins to be used in the calculation (minimum 4, maximum 144). More bins will generally give a more accurate result, but do not choose more bins than the number of signal samples in one revolution of the rotor.

8.4 Extreme predictions

Use this calculation to estimate lifetime extreme loads from a sample time history.

Click the **Select...**[8.20] buttons to define the signal to be processed and the signal representing rotor azimuth. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case. Then enter the following information:

- **Time spent at specified condition**: the total amount of time during the turbine lifetime for which the conditions of the sample time history are considered representative.
- **Number of points per segment** (power of 2, maximum 4096)
- **Percentage overlap**
- **Window** (select)
- **Remove trends** (check box)

See Section 8.5.1

Then choose either automatic or manual selection of data. Automatic selection is usually adequate, but if entered manually, the parameter required is:

- **Number of bins**: the number of azimuthal bins to be used in the calculation (minimum 4, maximum 144). More bins will generally give a more accurate result, but do not choose more bins than the number of signal samples in one revolution of the rotor.

8.5 Auto spectrum

This calculates the auto-spectral density (frequency spectrum) of a signal.

Click the **Select...**[8.20] button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case. Then enter the information required, as follows:

- **Number of points per segment** (power of 2, maximum 4096)
- **Percentage overlap**
- **Window** (select)
- **Remove trends** (check box)

See Section 8.5.1

8.5.1 Options for spectral analysis

All calculations involving spectral analysis use a Fast Fourier Transform technique with ensemble averaging. To perform the spectral analysis, the signal is divided into a number of segments of equal length, each of which contains a number of points which must be a power of 2. The segments need not be distinct, but may overlap. Each segment is then shaped by multiplying by a ‘window’ function which tapers the segment to zero at each end. This improves the spectrum particularly at high frequencies. A choice of windowing functions is available. Optionally, each segment may have a linear trend removed before windowing, which can improve the spectral estimation at low frequencies. The final spectrum is obtained by averaging together the resulting spectra from each segment.

The information required is therefore as follows:

- **Number of points**: the number of datapoints per segment. Must be a power of 2, maximum 4096. More points will give better frequency resolution, which may be important especially at low frequencies. However, choosing fewer points may result in a smoother spectrum because there will be more segments. If in doubt, 512 is a good starting point.

- **Percentage overlap**: the overlap between the segments. Must be less than 100%. 50% is often satisfactory, although 0% may be more appropriate if a rectangular window is used.
- **Window**: this may be
  (a) rectangular (equivalent to not using a window)
  (b) triangular:
  $$1-|2f-1|$$
  (c) Hanning:
  $$(1-\cos(2\pi f))/2$$
  (d) Hamming:
  $$0.54-0.46\cos(2\pi f)$$
  (e) Welch:
  $$1-(2f-1)^2$$
  Where f is the fractional position along the segment (0 at the start, 1 at the end). One of the last three windows (which are all quite similar) is recommended.
- **Trend removal**: Usually desirable.

8.6 Cross spectrum

This calculates the cross-spectral density, coherence and transfer functions between any two signals.

Click the **Select...**[8.15] buttons to define the two signals to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results for each load case will then be stored as additional outputs of that load case. Then enter the information required, as follows:

- **Number of points per segment** (power of 2, maximum 4096)
- **Percentage overlap**
- **Window** (select)
- **Remove trends** (check box)

$\Big\}$ See Section 8.5.1

8.7 Probability density

This calculation generates the probability distribution of a signal, and also, for comparison, the Gaussian or normal distribution with the same mean and standard deviation.

Using the **Revs at Level** option, it is also possible to bin the signal based upon a signal other than time. In this case the **Azimuth** signal will also need to be defined. If rotor azimuth is selected for this, the load level will be binned against the number of revolutions, which is useful in gearbox and bearing design.

Click the **Select...**[8.20] button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results will be accumulated over the turbine lifetime.

For each variable to be processed, enter the following data:

- **Minimum value**: the start of the distribution (should be less than the signal minimum)[8.12].
- **Maximum value**: the end of the distribution (should be greater than the signal maximum)[8.12].
- **Number of bins**: suggest 20. A higher number (maximum 144) will give finer resolution, provided there is sufficient data.
- **Remove mean** (check box): If this is selected, the distribution will be representative of deviations from the mean, and so the distribution mean will be zero. If Multiple Channels is selected, the means will not be removed.

8.8 Peak value analysis

This calculation generates the probability distribution of the peaks and troughs of a signal.

Click the **Select...**[8.20] button to define the signal to be processed. button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results will be accumulated over the turbine lifetime.

For each variable to be processed, enter the following data:

- **Minimum value**: the start of the distribution (should be less than the signal minimum)[8.12].
- **Maximum value**: the end of the distribution (should be greater than the signal maximum)[8.12].
- **Number of bins**: suggest 20. A higher number (maximum 144) will give finer resolution, provided there is sufficient data.

8.9 Level crossing analysis

This calculation generates the frequency with which a signal crosses given levels.

Click the **Select...**[8.20] button to define the signal to be processed. button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results will be accumulated over the turbine lifetime.

For each variable to be processed, enter the following data:

- **Minimum value**: the start of the distribution (should be less than the signal minimum)[8.12].
- **Maximum value**: the end of the distribution (should be greater than the signal maximum)[8.12].
- **Number of bins**: suggest 20. A higher number (maximum 144) will give finer resolution, provided there is sufficient data.

8.10 Rainflow cycle counting

This calculation generates the rainflow cycle count for a stress history. A suitable stress history can be generated from the relevant loads using the Channel combination[8.14] calculation.

Click the **Select...**[8.20] button to define the stress signal to be processed. button to define the signal to be processed. If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results will be accumulated over the turbine lifetime.

For each variable to be processed, enter the following data:

- **Minimum value**: the start of the stress distribution (should be less than the minimum stress)[8.12].
- **Maximum value**: the end of the stress distribution (should be greater than the maximum stress)[8.12].
- **Number of bins**: suggest 20. A higher number (maximum 128) will give finer resolution, provided there is sufficient data.
- **Minimum range**: the smallest signal range to be counted as a cycle. A non-zero value may be useful to remove the effect of any spurious noise on the signal.

If you select **Calculate Equivalent Loads**, enter up to 10 inverse S-N slopes to use for the calculation of equivalent loads. These are the sinusoidal loads which would produce the same fatigue damage at the specified S-N slope if the frequency of the sinusoid is as specified.

8.11 Fatigue damage estimation

This calculation generates fatigue damage estimates from a stress history or a previously generated rainflow cycle count, by taking account of the fatigue properties of the material.
A suitable stress history can be generated from the relevant loads using the Channel combination[8.14] calculation.

Click the **Select...**[8.20] button to **either** the stress signal to be processed, **or** a previously generated rainflow cycle count[8.10] (not available for Multiple Channels). If you have selected Multiple Channels[8.15] you will be able to specify a number of load cases and variables to be processed in a single calculation, and the results will be accumulated over the turbine lifetime.

Then enter the material properties as follows:

Fatigue model

- **S-N curve**
  Select either the **log-log** or the **look-up table** option. The fatigue properties are then entered as follows:
  If **Log-log relationship** is selected: enter the **inverse slope** (m) and **intercept** (c) of the S-N curve, such that the stress range (S) giving N cycles to failure is given by:
  $\log(S) = \log(c) - (1/m) \log(N)$

If **Look-up table** is selected, click the **define fatigue data look-up table button** for a pop-up window. Use **Add** or **Insert** to add points to the look-up table, and enter the stress values and the corresponding number of cycles to failure by double-clicking on the appropriate table entries. The stress entries must be monotonically decreasing, with the corresponding cycles to failure monotonically increasing.

- **Goodman correction**
  Use the check box to enable the Goodman correction if required. If required, enter the ultimate strength of the material, as a stress.

If a **stress history** rather than a previously generated rainflow cycle count is to be processed, enter the following data which is required for the cycle counting:

- **Minimum value**: the start of the stress distribution (should be less than the minimum stress)[8.12].
- **Maximum value**: the end of the stress distribution (should be greater than the maximum stress)[8.12].
- **Number of bins**: suggest 20. A higher number (maximum 128) will give finer resolution, provided there is sufficient data.
- **Minimum range**: the smallest signal range to be counted as a cycle. A non-zero value may be useful to remove the effect of any spurious noise on the signal.

8.12 Setting bin limits

Several calculations require minimum and maximum values to be entered for the bin range. These values should encompass the entire data range.

For automatic calculation of suitable bin limits, set the minimum and maximum to be equal, or leave them both blank.

Alternatively, the actual minimum and maximum of the signal may be found by running the Basic statistics[8.1] calculation.

8.13 Annual energy yield

This calculation generates the annual energy yield of a wind turbine from either a steady state power curve[7.8], a dynamic power curve[7.14] or a series of power production simulations, as a function of annual mean wind speed, assuming either a Weibull, Rayleigh or a user defined distribution of wind speeds[6.11].

Click the **Select...**[8.20] button to define the power curve to be used. Then enter the following data:

- **Minimum wind speed**: the cut-in wind speed for the turbine
- **Maximum wind speed**: the cut-out wind speed for the turbine

- **Wind turbine availability**: the average availability, assumed to be uncorrelated with wind speed.

Define the wind speed distribution[6.11] by clicking the **Define** button. If desired, select the option to scale the wind speed distribution by a series of factors; the energy yield will then be calculated as a function of annual mean wind speed while retaining the shape of the wind speed distribution. In this case, enter

- **Starting annual mean wind speed**
- **End annual mean wind speed**
- **Annual mean wind speed step**

Energy yield will be calculated for each mean wind speed in this range

8.14 Channel Combination

Use this calculation to combine and scale a number of signals. This is particularly useful for generating a stress history for a particular point in the wind turbine structure, by expressing it as a linear combination of the various loads acting on the component. The stress history can then be used to calculate fatigue[8.11] damage. The scale factors used for the linear combination should be such that the resulting signal is in the same stress units as are assumed in the fatigue damage calculation.

8.14.1 Multiple processing option

If **Combine variables across different load cases** is not selected, a calculation can be set up in which up to 18 signals from a particular load case may be combined to generate one or more combined signals attached to that load case. A list of load cases may be specified, so that the same set of channel combinations is repeated for each of the load cases.

First define the **Combined Signal File No** (from 1 to 260; up to 260 combined signal files may be generated for any load case), and enter a description for the file. To load the details of an previous Channel Combination calculation, click **Import details…** and select whether to import the channel combination details, the list of load cases, or both.

Click **Channels and Load Cases** to edit or define the channel combination details and the list of load cases. Up to 18 input signals can be combined, to generate anything up to 50 output signals. Click **New >>** to define a new output signal, and enter a description for it. Select one of the available units for the signal if appropriate.

For each output variable, click **Add Variable…** to select the required input variables. Specify any factor, offset and other unary operators to be applied to each input signal before it is combined with the other input signals. The Factor is applied first, then the Offset, then any Unary operators are applied. If several unary operators are specified, they must be separated by | (a vertical bar). They will be applied in order, starting from the right-most. Allowed operators are:"

SIN	sine (of a value in radians)
COS	cosine (of a value in radians)
ABS	absolute value
SQRT	square root

INV	reciprocal
+n	Add a number n
-n	Subtract n
xn	Multiply by n
/n	Divide by n
^n	Raise to the power n

where n is a real number, which may be negative. For example, **SIN|x-3.2|^-0.2|ABS|+0.1** would calculate $\mathbf{sin(-3.2(|\boldsymbol{x}+0.1|^{-0.2}))}$ from the input signal $\boldsymbol{x}$.

Then select how the resulting input signals are to be combined. This may be by

- addition,
- squaring and adding, then taking the square root of the result, or
- multiplying.

Finally click **Load Cases** to set up a list of load cases for which the resulting set of channel combinations will be repeated.

8.14.2 Single channel combinations

Alternatively, select **Combine variables across different load cases** to generate a new variable not attached to any existing load case. You will then be asked for a directory and run name for the output when you start the run. In this case, click the numbered channel selection[8.20] buttons to define up to six signals to be combined. The signals may be from different runs or load cases if desired. For each signal, enter a scale factor and an offset. Enter a description for the combined signal, and choose units if appropriate.

The scale factors (a_i) are applied before the offsets (b_i), i.e. the result of the calculation can be expressed as:

$$y = \sum \left(a_i x_i + b_i\right)$$

if **Simple Addition** is selected. If **Square-Add-Square Root** is selected, the result is

$$y = \sqrt{\sum \left(a_i x_i + b_i\right)^2}$$

Enter a description for the combined signal, and select one of the available units for the combined signal if appropriate.

8.15 Multiple Processing

A number of post-processing calculations allow a whole list of variables to be processed across a whole list of load cases. If appropriate, the results are accumulated over the turbine lifetime (or any other desired period) assuming a particular wind speed distribution, and a frequency of occurrence for transient events. Otherwise the results are stored as additional outputs for each load case.

In the window for the required post-processing calculation, click **Multiple Channels** to bring up the **Multiple Processing** window. Use the **Add Load Case** and **Add Variable** buttons to build up the required lists of load cases and variables. **Add Many Load Cases** allows all the calculations in a selected directory (including subdirectories if desired) to be added in one go. In most cases, multiple variables can be selected in one go by dragging over a number of variables, or holding down the **Control** key and clicking on the required variables.

It is important to ensure that the output[7.19] specifications for all the load cases are the same, i.e. the same outputs and the same blade and tower stations have been requested.

If available, in the **Load Cases** window, enter the Occurrences per Year for transient load cases such as stops and starts, or the wind speed bin or number of hours per year which the load case is deemed to represent in the case of stationary load cases; click the **↔** button to select either wind speed bins or number of hours per year. It is possible to treat a stationary load case as transient, or vice versa, by clicking on the **'S'** or **'T'** indicator. At the bottom, enter the turbine lifetime, and ensure a wind speed distribution[6.11] is defined. The wind speed distribution parameters will not be used if all the load cases are transient, and the length of transient simulations will not be included in the turbine lifetime.

If available, for each variable, enter the number and range of the bins to be used. If the Minimum and Maximum are both left as zero, a bin range will be calculated which is suitable across all the load cases. For rainflow cycle and fatigue analysis, it is possible to specify a minimum range below which cycles are not counted.

8.16 Ultimate Loads

This calculation scans a number of load cases to find the maximum and minimum values of a specified set of variables, and records the simultaneous values of all the other variables in the list.

From the **Ultimate Loads** window, click **Define Channels and Load Cases** to open the Multiple Processing window. Use the **Add Load Case** and **Add Variable** buttons to build up the required lists of load cases and variables.

For users with the **Advanced Post Processing** module, a list of **Load Case Groups** can be set up along with their corresponding safety factors. A check box on the **Load Cases** tab enables these to be assigned to the load cases. During processing the safety factors will then be applied. Safety factors may also be assigned to individual variables, and these will override the group safety factor.

Each load case should have a unique identifier. If **Load Case Groups** have been defined, the identifier will attempt to set itself to a sensible default, which the user is free to change if required. For each variable, specify whether its maximum and minimum are to be found.

An additional feature available to users with the **Advanced Post Processing** module is to combine blade loads across all blades. With this option the maximum and minimum loads across all blades will be reported.

The output from this calculation consists of an ASCII text file with many rows and columns. The file name will be <directory> \ <run name> .$mx where the directory and run name are entered at the time of running the calculation. The file is suitable for importing directly into a spreadsheet, and can also be tabulated using the **Multiple Plots**[9.12] feature. For users with the **Advanced Post Processing** module, additional output is available in the form of histogram plots, which present comparison between the load cases group. A drop down list on the **Load Cases** tab allows the desired type of histogram output to be selected.

You can also use the Ultimate Load Cases calculation to process one or more Ultimate Loads output files and give a graphical presentation of results.

8.17 Ultimate Load Cases

This calculation generates a list of maxima and minima of loads and the load cases in which they occur.

In the **Ultimate Load Cases** window, click **Add...** to select output files from ultimate loads calculations. Select the required variables from the list of available variables.

To view the results, click the **Data view** icon on the main toolbar and select the **Special** graph type. Then find the calculation results by clicking **Channel 1**. The results will be presented as a histogram.

8.18 Flicker

Flicker is a measure of the severity of voltage fluctuations caused by equipment (such as a wind turbine) connected to the network. It may be calculated either from a voltage time history or from active and reactive power. Click the **Select...** button(s)[8.20] to define the time histories to use.

Range of data to use: if desired, flicker may be calculated from just a part of the time history. To do this, enter a **To** value which is greater than the **From** value.

If flicker is calculated from active and reactive power, it can be calculated for a range of different network short-circuit power levels and impedance angles in a single calculation. Enter as many values as are required. **Set default values** supplies a set of standard impedance angles.

See also:
Generator[4.8], Network[4.10].

8.19 Linear Model

This calculation is available for users with a licence for the Linearisation module. It converts the output of a **Model Linearisation**[7.11] calculation into a state-space model, in a form which is suitable for controller design and is directly compatible with Matlab [2]. A state space model has the following form:

$$\dot{x} = Ax + Bu$$
$$y = Cx + Du$$

where u is a vector of inputs, x is a vector of system states, and y is a vector of outputs. A, B, C and D are the state space matrices.

First click **Select…** to define which variables from the **Model linearisation** results are required as model outputs. The model states will depend on the dynamics which were selected in the wind turbine model for that calculation, and the inputs will include wind speed, pitch angle demand and generator torque demand. The state space matrix coefficients are calculated as the slope of a best fit line through a number of points generated by perturbations of different sizes away from the steady state condition.

The **Minimum acceptable correlation coefficient** defines whether a best fit will be accepted or not. If the correlation is poorer than this, that particular matrix coefficient is set to zero. A value of 0.5 is generally suitable.

Two output files are created in the selected output directory, using the selected run name and the following file extensions:

<RunName>.mat	This is a '.mat' file suitable for reading directly into Matlab [2].
<RunName>.$m2	This is an ASCII text file, also suitable for reading into Excel, containing the same information.

Each file contains the four state space matrices for each operating point, and also wind speeds and rotor azimuths defining the operating points. There are also vectors containing the steady state values of all inputs, states and outputs at each operating point, as well as some additional information which may be of use in controller design such as the gearbox ratio, number of blades, and nominal speed and torque values.

In the .mat file, the variable names used are fairly self-explanatory, except that the state space matrices and the names of the inputs, states and outputs are all stored in a structure called SYSTURB. The elements of SYSTURB are:

inputname	The names of the input variables (character array)
outputname	The names of the output variables (character array)
statename	The names of the state variables (character array)
A, B, C, D	Arrays of the state-space matrices for the different operating points; thus A(i,j,k,l) is the i,jth element of the A matrix (i.e. row I, column j), for the kth wind speed and the lth rotor azimuth angle.

This structure can readily be converted to a Matlab lti model array using the single Matlab command:

```
SYS = ss (SYSTURB.A, SYSTURB.B, SYSTURB.C, SYSTURB.D, …
        'inputname', cellstr(SYSTURB.inputname), …
        'outputname', cellstr(SYSTURB.outputname), …
        'statename', cellstr(SYSTURB.statename));
```

Diagnostic plots may be generated for all matrix coefficients, or for all matrix coefficients whose correlation coefficients fall within a specified range of values. **Note**: Diagnostic plots cause a much slower calculation, and may generate very large numbers of plot files; these are stored as enhanced metafiles in the selected run output directory, with the following naming convention:

<RunName><M><W>Az<A>_<row>_<col>_<?><cc>.emf

<RunName>	the run name selected for the Linear Model calculation
<M>	which state-space matrix: A, B, C or D the plot refers to
<W>	the operating point wind speed
<A>	the rotor azimuth angle
<row>, <col>	which particular coefficient of the matrix the plot refers to
<?>	OK for a coefficient which was accepted, X for one which was rejected
<cc>	the correlation coefficient obtained for the best fit line.

8.20 Data channel selection

This section describes the procedure for selecting calculation results for viewing or post processing.

Click the channel selection button on either the graph parameters[9] screen (for viewing data), or on any of the post processing[8] screens. Then take the following steps to find the data you require:

1. Select the disk drive and the directory where the data is to be found using the selection boxes provided.
2. Select the run name which identifies the calculation results.
3. Click on the **Data group** containing the required data. The **Variables** box below will show the data available in this group. In some situations, it is possible to select a number of data groups simultaneously.
4. When the correct group has been selected, select the desired variable in the box below. Again in some situations, it is possible to select a number of variables simultaneously.
5. If appropriate, select the independent variable as described below.

8.20.1 Selecting independent variables

Independent variables are shown in the box at the bottom of the channel selection screen. The number of independent variables required depends on the context. For example, a line graph or histogram requires one independent variable (shown as X-Axis), and a 3D graph requires two (shown as X-Axis and Z-Axis). For post-processing, the last independent variable (usually time) is selected by default and is not listed. Where there are more independent variables than required, particular values must be selected for the remaining variables, by clicking on the variable and selecting a value in the box on the right. Double-click on a variable to change whether or not it will be used as an independent variable.

For example, if the independent variables are **Blade station radius** and **Time**, you may click on **Blade station radius** and select the radius required, leaving **Time** as the X-Axis for the graph. Alternatively, double-click on **Time** to select a particular value of time, and you can then plot the variable at that time as a function of blade station radius. If you make both variables independent, you will get a 3D graph.

8.20.2 Messages and further information

The **View Messages** button displays any warning or error messages generated during the run. In a few cases, some additional information is available by clicking the **View Info** button.

8.20.3 Deleting information

Buttons are available on the data channel selector to permanently delete a data group or an entire run.

9. GRAPHICAL DATA VIEW FACILITY

Click the **Data view** icon on the toolbar to open the **Graph Parameters** screen.

First click the selection button for either a two-dimensional or three-dimensional graph. Then click the **Channel 1** button to select the data you wish to view - see Data channel selection[8.15] for details of selecting data for two- or three-dimensional graphs. A **Special** graph type is available for results of the Ultimate Load Cases[8.17] calculation.

If required, make any changes to the format of the graph on the **Graph Parameters** screen. Then click the **View Graph** button to display the graph. The graph can then be printed if desired, using the **Print** button, or saved to a metafile or bitmap using the **Save** button. The metafile can then be inserted into a report[9.10] . Alternatively, copy the graph to the clipboard using the **Copy Metafile** or **Copy Bitmap** button.

The selected graph data may also be tabulated by clicking the **Tabulate** button[9.9]. Select whether to view the tabulated data on screen or send it to an ASCII file, a WORD document or an EXCEL spreadsheet.

Click **Stats** to tabulate the mean, minimum, maximum and standard deviation of the selected datasets.

9.1 Graphs of several variables

In the case of two-dimensional graphs, it is possible to plot up to six datasets on one graph. Simply click additional channel selection buttons to define the data required. Click the **Clear** buttons to remove unwanted channels. Any channel may also be disabled temporarily by means of the check-box on the right hand side.

9.2 Graph styles

Two-dimensional graphs may be displayed as line graphs, points or bar charts by clicking the appropriate Style button on the **Graph Parameters** screen. In the same way, three-dimensional graphs may be displayed as surface plots or column plots.

9.3 Grids and logarithmic axes

Use the check-boxes on the **Graph Parameters** screen to specify whether or not to add X and Y grids to the graph, and whether or not to plot on logarithmic axes in the X and/or Y directions. Logarithmic axes are only available on two-dimensional graphs.

The **Darker grids** option may be useful for printing the graph to some black and white printers. The feint grid lines may not always appear, depending on the printer driver.

9.4 Units

A **units** selection box to the right of each channel allows a choice of units for the variable being plotted, where appropriate. Two selection boxes are available for the case of cross-spectra and transfer functions with composite units.

Selection boxes in the lower part of the **Graph Parameters** screen give a choice of units for the X-axis where appropriate, and for the Z-axis in the case of three-dimensional graphs.

9.5 Axis limits

The default axis limits shown in the lower part of the **Graph Parameters** screen may be changed if desired, to alter the limits on the X and Y axes, and also on the Z axis in the case of three-dimensional graphs. The default tick interval may also be changed.

Clicking on an entry with the Shift button depressed will freeze it so that it will no longer be updated automatically as channel selections and units are changed. A second shift-click will unfreeze the entry, so that it will be automatically updated.

9.6 Graph titles

The default axis labels (X, Y and Z if appropriate) and graph titles (top and bottom) may be edited in the lower part of the **Graph Parameters** screen. Use a tilde (~) to start a new line in the graph title.

Clicking on an entry with the Shift button depressed will freeze it so that it will no longer be updated automatically as channel selections and units are changed. A second shift-click will unfreeze the entry, so that it will be automatically updated.

9.7 Graph legends and line styles

For two-dimensional graphs, a legend may be added. This is useful for plots showing more than one dataset, as it allows the different datasets to be identified. Click the **Legends and line styles** button to edit the default legend captions and to specify where the legend is to appear on the graph. One of five line thicknesses may be selected for each line of the graph.

Clicking on a legend entry with the Shift button depressed will freeze it so that it will no longer be updated automatically as channel selections and units are changed. A second shift-click will unfreeze the entry, so that it will be automatically updated.

9.8 Cross-plots

Two dimensional graphs can be presented as cross plots if two or more channels with compatible independent variables have been selected. Use the **X-axis** selector to define which channel is to be plotted along the X axis. The other channels will then be plotted against this one.

9.9 Tabular output of results

Click the **Tabulate** button on the **Graph Parameters** screen to tabulate the current graph data. A dialogue box allows you to select the format:

Screen: the data can be viewed on the screen.
ASCII: to save the table as tab-delimited ASCII text.
Excel: to save the table as a Microsoft Excel workbook (You must have Microsoft Excel installed)
Word: (the default) to save the table in a Microsoft Word document (you must have Microsoft Word installed).

When viewed on the screen, the data may also be copied into the clipboard for transfer to another application. This is sometimes quicker than sending the table directly to EXCEL. For a large table, it is sometimes faster to send it to an ASCII file first and then read the file into EXCEL.

9.10 Refreshing graphs

Click **Refresh Data** to re-read any plot data which has changed since it was last read in, for example in the case of a plot of results from a simulation which is still in progress, or if a calculation has been repeated, overwriting the old results. Any channel can be forcibly re-read using the **Data** pull-down menu.

Select the **Auto-refresh graph every ...** option to re-read the graph data at the specified time intervals. This is useful for a graph to update itself automatically as a simulation progresses. If the interval is set to zero, the graph will not update, the **View** button will now use the same graph window each time it is pressed rather that opening a new window each time.

From the pull-down menu, use **Data -> Change runs** to read new graph data from one or more different runs, for the same variables.

9.11 Graph configurations

The **File** pull-down menu allows you to save and load **Bladed Graph Configurations** (.bgc files) which contain all the necessary details to re-create a graph (the graph size, the channel definitions, styles, and also axis limits and graph annotation whether fixed or automatic). When loading a configuration, a prompt allows the run(s) and variable(s) to be changed. This provides a useful and flexible way of storing and re-using a particular graph configuration for different sets of results.

9.12 Multiple Plotting and Tabulation

Click the **Multiple Plots** button to bring up the Multiple Plotting screen. This feature allows a graph configuration file to be used as a template for a graph or table, with the variable in the graph or table being replaced from lists of variables and runs. Clicking the **Start** button will start the series of plots in the order specified by the **sequence control**. This facility is only available to users with the **Advanced Post Processing** module. The graphs and tables may be viewed on screen and/or sent directly to a report.

9.12.1 Graph configurations

Add one or more graph configurations[9.11] to the list, specifying whether any of the runs and/or variables defined in the graph configuration are to be replaced from the lists below.

9.12.2 Replacing variables

If any variables in the graph configuration(s) are to be replaced, they will be replaced by each item in the **Variables** list in turn. **Add** any desired variables to the list, or use **Add Set** to process complete sets of a specific type of graph or table. Clicking on the **Add Set** button will bring up a choice of the type of set required. All the data in a particular run that is part of the specified set will then be processed. The following sets are available:

- Ultimate Loads (histogram output)
- Rainflow Cycle Exceedance
- Rainflow Cycle By Range
- Lifetime Weighted Equivalent Loads

9.12.3 Replacing runs

If any runs in the graph configuration(s) are to be replaced, they will be replaced by each item in the **Runs** list in turn. **Add** any desired runs to the list; click **Add Many** to add all the runs in a directory or a directory tree.

9.12.4 Sequence Control

The sequence control allows the user to specify the order in which the plots or tables will be generated.

9.12.5 Tabulation of Ultimate Loads

Select the **Tabulation of Ultimate Loads** to tabulate the Ultimate Loads from each of the ultimate loads calculations specified in the **Runs** list. There is no graphical output available in this case.

9.12.6 Output

Use the options in the **Output Requirements** to specify what type of output is required, and where it should be sent. Graphs can be displayed on screen, stored as metafiles, and sent to a WORD document. Tabulated data and statistical data can also be sent to a WORD document or to an ASCII text file. If an output file is specified which already exists, data will be appended to the end of the file.

10. REPORTING

The ***Bladed*** reporting system enables the user to generate reports describing the wind turbine characteristics and the details of any particular calculation. Calculation results can also be entered into the report in tabular or graphical form. The report is prepared directly in Microsoft Word format. Alternatively, ASCII text format is available as an option if required, but this clearly does not allow the inclusion of graphical output.

From the **Reports** menu on the main toolbar, select the required report format from the following options:

ASCII: to save the table as tab-delimited ASCII text.
Word: (the default) to save the table in a Microsoft Word document (you must have Microsoft Word installed).
The current format can be saved as the default using the **Save** item of the **Options** pull-down menu on the main toolbar.

There are two main ways of generating reports:

- **Project reports**: these may contain any currently assigned modules defining the wind turbine itself, as well as calculation parameters and wind field details if required.

- **Calculation reports**: these may include any turbine details which are currently assigned and which are relevant to the calculation, as well as details of the calculation itself and the wind field used if applicable. They may be generated as stand-alone reports, or appended to another report such as a project report.

As well as adding graphs and tables, the reports may also be edited and printed within ***Bladed.***

10.1 Project reports

From the **Reports** menu on the main toolbar, select **Write project report.** A dialogue box allows a document name to be chosen. If it is an existing document, you may choose to overwrite it or to append to it.

The **Report Contents** screen then appears. Select the desired report contents by checking or unchecking the appropriate boxes. By default, this includes any currently assigned modules defining the wind turbine itself. Calculation parameters and wind field details may also be added. Click on **OK** to generate the report.

10.2 Calculation reports

Calculation reports can be written for any calculation which has been performed with a defined run name. From the **Reports** menu on the main toolbar, select **Write calculation report.** A dialogue box allows a particular run to be chosen.

The **Report Contents** screen then appears. Select the desired report contents by checking or unchecking the appropriate boxes. Select the desired report contents by checking or unchecking the appropriate boxes. If desired, change the **Destination** selector to **Other** to change the report file name. You may append to an existing report if desired. Click on **OK**. to generate the report. If an **Other** destination was selected, a dialogue box appears to allow you to select an appropriate file name

10.3 Adding calculation results to a report

When viewing a graph[9], click the **Save** button to save the graph to a metafile. Then, or at any later time, go to the **Reports** menu on the main toolbar and select **Append Graph.** Dialogue boxes appear, allowing you to select the metafile required and the Word document to which it will be appended. The graph may be inserted fully or as a link - see Linked graphs[10.5].

Alternatively, click **Copy Metafile** on the graph window to copy the graph to the clipboard, and paste it directly into the report.

Graphs cannot be appended to ASCII reports.

To append a table of results rather than a graph, refer to Section 9.9.

10.4 Editing and printing reports

An existing report may be edited and printed. From the **Reports** menu on the main toolbar, select **Edit report** or **Print report**.

If **Edit report** is selected and the report format is set to **Word**, Microsoft Word will be opened and any desired editing may be carried out. If the report format is ASCII, the Windows Notepad editor will be opened.

10.5 Linked graphs

From the **Tools -> Preferences** menu on the main toolbar, the **Insert graphs as links** option allows graphics inserted into a Word report using the **Reports** pull-down menu option to be linked to an external metafile. The Word document will be smaller, but not transportable unless the metafile is moved with it.

APPENDIX A Communication Between *Bladed* And External Controllers

This appendix describes in detail the communication between ***Bladed*** and the user's external controller code. See also Section 5.8.

Appendix B gives some simple examples of external controller code in several languages.

A.1 Data exchange records

External controllers compiled as executable (.EXE) files exchange information with the ***Bladed*** simulation through a shared binary file consisting of a number of 4-byte records. External controllers compiled as DLLs exchange information through an array passed as the first argument to the DLL. The structure of the binary file used for EXEs and the array used for DLLs is similar and is described in the tables which follow. The type of each record of the file or element of the array may be integer, real or character, as specified in the tables. In the EXE case, the 4-byte records in the file should be written to and/or read in as 4-byte integers, 4-byte (single precision) real (i.e. floating point) numbers, or groups of 4 characters as appropriate. In the DLL case, all the array elements are passed as real numbers, so if an element is described as type Integer, the real number must be converted to the nearest integer (and integers being sent back to the simulation must be converted to real values). Character variables are passed in separate arrays in the DLL case, as described in Section 5.8.2.

Table A.1 shows the array elements or binary file records which are used for data exchange between the ***Bladed*** simulation and the external controller. As shown by the 'Data flow' column, some records are used to pass information from the simulation to the controller, some are used to pass information from the controller back to the simulation, and a few are used for two-way communication.

Note that the first binary file record or array element is referred to as record or element number 1 (not 0).

Record number	Data flow[8]	Data type[9]	Description	See note(s)	Units
1	in	I	See Section A.2		-
2	in	R	Current time		s
3	in	R	Communication interval		s
4	in	R	Blade 1 pitch angle		rad
5	in	R	Below-rated pitch angle set-point	1	rad
6	in	R	Minimum pitch angle	1	rad
7	in	R	Maximum pitch angle	1	rad
8	in	R	Minimum pitch rate (most negative value allowed)		rad/s
9	in	R	Maximum pitch rate		rad/s
10	in	I	0 = pitch position actuator, 1 = pitch rate actuator		-
11	in	R	Current demanded pitch angle		rad
12	in	R	Current demanded pitch rate		rad/s
13	in	R	Demanded power	2	W
14	in	R	Measured shaft power	3	W
15	in	R	Measured electrical power output		W
16	in	R	Optimal mode gain	3,5	$Nm/(rad/s)^2$
17	in	R	Minimum generator speed	3	rad/s
18	in	R	Optimal mode maximum speed	3	rad/s
19	in	R	Demanded generator speed above rated	1,3	rad/s
20	in	R	Measured generator speed		rad/s
21	in	R	Measured rotor speed		rad/s
22	in	R	Demanded generator torque	3	Nm
23	in	R	Measured generator torque	3	Nm
24	in	R	Measured yaw error	4	rad
25	in	I	Start of below-rated torque-speed look-up table =R	3,5	Record no.
26	in	I	No. of points in torque-speed look-up table =N	3,5	-
27	in	R	Hub wind speed	4	m/s
28	in	I	Pitch control: 0 = collective, 1 = individual		-
29	in	I	Yaw control: 0 = yaw rate control, 1 = yaw torque control		-
30-32	in	R	Blade 1-3 root out of plane bending moment	18	Nm
33	in	R	Blade 2 pitch angle		rad
34	in	R	Blade 3 pitch angle		rad
35	both	I	Generator contactor	10	-
36	both	I	Shaft brake status: 0=off, 1=on		-
37	in	R	Nacelle angle from North		rad
38-40	out		Reserved		
41	out	R	Demanded yaw actuator torque	2	rad
42	out	R	Demanded blade 1 individual pitch position or rate	12,14	rad or rad/s
43	out	R	Demanded blade 2 individual pitch position or rate	12,14	rad or rad/s
44	out	R	Demanded blade 3 individual pitch position or rate	12,14	rad or rad/s
45	out	R	Demanded pitch angle (Collective pitch)	12	rad
46	out	R	Demanded pitch rate (Collective pitch)	12	rad/s
47	out	R	Demanded generator torque		Nm
48	out	R	Demanded nacelle yaw rate	13	rad/s

....continued overleaf....

....continued....

Record number	Data flow[8]	Data type[9]	Description	See note(s)	Units
49	out	I	Message length OR $-M_0$	15	-
49	in	I	Maximum no. of characters allowed in the "MESSAGE"	6	-
50	in	I	No. of characters in the "INFILE" argument	6	-
51	in	I	No. of characters in the "OUTNAME" argument	6	-
52	in	I	DLL interface version number (reserved for future use)	6	-
53	in	R	Tower top fore-aft acceleration		m/s^2
54	in	R	Tower top side to side acceleration		m/s^2
55	out	I	Pitch override	16	-
56	out	I	Torque override	16	-
57-59	out		Reserved		
60	in	R	Rotor azimuth angle		rad
61	in	I	No. of blades		-
62	in	I	Max. number of values which can be returned for logging	7	-
63	in	I	Record number for start of logging output	7	-
64	in	I	Max. no. of characters which can be returned in "OUTNAME"	7	-
65	out	I	Number of variables returned for logging	17	-
66-68	in	R	Reserved		
69-71	in	R	Blade 1-3 root in plane bending moment		Nm
72	out	R	Generator start-up resistance		ohm/phase
73	in	R	Rotating hub My (GL co-ords)	18	Nm
74	in	R	Rotating hub Mz (GL co-ords)	18	Nm
75	in	R	Fixed hub My (GL co-ords)	18	Nm
76	in	R	Fixed hub Mz (GL co-ords)	18	Nm
77	in	R	Yaw bearing My (GL co-ords)	18	Nm
78	in	R	Yaw bearing Mz (GL co-ords)	18	Nm
79	out	I	Request for loads	18	-
80	out	I	1 = Variable slip current demand at position 81	11	-
81	both	R	Variable slip current demand	11	A
82	in	R	Nacelle roll acceleration	18	rad/s^2
83	in	R	Nacelle nodding acceleration	18	rad/s^2
84	in	R	Nacelle yaw acceleration	18	rad/s^2
R	in	R	First generator speed in look-up table	3,5	rad/s
R+1	in	R	First generator torque in look-up table	3,5	Nm
R+2	in	R	Second generator speed in look-up table	3,5	rad/s
R+3	in	R	Second generator torque in look-up table	3,5	Nm
...		...	... etc., until ...		...
R+2N-2	in	R	Last generator speed in look-up table	3,5	rad/s
R+2N-1	in	R	Last generator torque in look-up table	3,5	Nm
M_0	out	I	Message length, only if record 49 < 0	15	
M_1 - M_n	out	C	Message text, 4 characters per record	15	-
L_1 onwards	out	R	Variables returned for logging output	17	SI

....continued overleaf....

....continued....

Notes:
1. Pitch regulated case only.
2. Not for variable speed pitch regulated case.
3. Variable speed case only.
4. Based on free wind at hub position - no modelling of actual nacelle anemometer or wind vane.
5. If the look-up table option is selected for the optimal mode below rated control, then record 16 is zero, record 25 contains the record number (R) of the start of the look-up table, and record 26 contains the number of points in the table (N).
6. DLL case only: see Sections 5.8.2 and A.3.
7. DLL case only: see Section A.5.
8. in = data supplied by simulation, which may be used but not changed by the external controller.
   out = data supplied by the external controller to the simulation.
   both = data which is written by the simulation but which may be changed by the external controller.
9. Record type for EXE case. I = integer, R = real (floating point), C = character. In the DLL case, all records are actually passed as 4-byte real (floating point) numbers.
10. 0 = off, 1 = main (high speed) or variable speed generator, 2 = low speed generator.
11. Only used with the variable slip generator electrical model. Set record 80 to 1 if using record 81 to send a rotor current demand. If record 80 is 0 (default), then the torque demand (record 47) will be used to control the generator.
12. See record 28.
13. See record 29.
14. Depending on record 10.
15. EXE case only: see Section A.3.
16. See Section A.4.
17. DLL case only; see Section A.5.
18. Record 79 is used to request additional measured loads and accelerations to be provided by the simulation:

Record 79	Blade loads and accelerations	Hub rotating loads	Hub fixed loads	Yaw bearing loads
0				
1	√			
2	√	√		
3	√	√	√	
4	√	√	√	√

Table A.1: Communication records

Note the strict use of SI units for all variables.

Note also that many of the parameters passed from the simulation to the controller are constants as defined in the **Control Systems** window, and some are variables such as measured signals. Some are only relevant for certain types of controllers, e.g. fixed or variable speed, stall or pitch control, and pitch position or pitch rate actuators. Although the record numbers are always the same, as shown in the tables above, the user-defined controller need only make use of those parameters which it actually requires, and only needs to output the demands which are relevant for the particular case, e.g.:

- demanded pitch angle(s) for pitch regulated machines with pitch position actuator
- demanded pitch rate(s) for pitch regulated machines with pitch rate actuator
- demanded generator torque for variable speed machines
- demanded nacelle yaw rate if the external controller option was selected for active yaw with yaw rate control
- demanded yaw actuator torque if yaw torque control was selected.

The controller may, if desired, change the status of the generator contactors and the brake.

A.2 Record 1: the Status flag

In the EXE case, record 1 of the shared binary file is used for handshaking, as described in Section 5.8.1.

In the DLL case, element 1 of the "DATA" array is set by the simulation as follows:

0	First call at time zero
1	All subsequent timesteps
-1	Final call at the end of the simulation.

The DLL may set the value to –1 to request the simulation to terminate.

A.3 Sending messages to the simulation

The controller may send a message to the simulation, which will then be displayed to the user.

In the DLL case, a separate argument to the DLL is provided for this purpose, as described in Section 5.8.2. Element 49 of the "DATA" array gives the maximum number of characters allowed. Each 1-byte element of the "MESSAGE" array can store one character of the message.

In the EXE case, there are two methods of specifying the message, which should not exceed 80 characters in length:

Method 1 (obsolete): Record 49 should contain the number of characters in the message, and the subsequent records should contain the message, four characters per record.

Method 2 (recommended): Place the message in records M_1 onwards, 4 characters per record. Enter the number of characters in the message as an integer in record number M_0 where $M_0 = M_1 - 1$, and set record 49 to $-M_0$ (note negative sign). Choose M_0 so that all these records occur after other output records, for example $M_0 = 61$. In practice it does not actually matter if any of the records in Table A.1 are overwritten since they are refreshed each timestep.

The EXE controller **must** write to record 49: a zero should be written if there is no message.

A.4 Pitch and torque override

If the external controller is used for supervisory control actions such as starts stops, while the built-in continous-time PI controllers are used for power production control, then it may be necessary for the external controller to specify the instant at which the supervisory control action takes over from the in-built controller action. Set record or element 55 to integer 0 whenever the external controller is to control the pitch, overriding the built-in PI controller. Set it to 1 when the built-in PI controller should be controlling the pitch.

For variable speed turbines, use record 56 in the same way to determine whether the external or the built-in controller should be controlling the generator torque.

Note that in the EXE case, messages should be written using Method 2 (see Section A.3) if the override control is to be used. The external controller will always take precedence if Method 1 is used.

A.5 Sending logging output to Bladed

In the DLL case only, additional data may be sent back to ***Bladed*** for logging in additional simulation output files in a similar format to other simulation outputs. This data can then be viewed directly using the Data View[9] facility, or post-processed[8]. This is particularly useful for debugging the controller, or for illustrating the details of its operation.

Element 62 of the "DATA" array gives the maximum number of logging outputs which can be returned. On the first call, the DLL should set element 65 to the number of logging outputs which will be returned, and their values should be returned starting at the element whose number is given by the value of element 63.

The "OUTNAME" array can be used to specify the names and units for the logging outputs. This should be set on at least the first and last calls to the DLL (overwriting the existing information in that array). This array should be set to a sequence of characters as follows:

Name:*Units*;

repeated for each logging output. *Name* is a description of the logging output, and *Units* should be one of table A.2, provided the logging output is presented in strict SI units.

See Appendix B1 for an example in 'C'.

Allowed values for *Units*	**Meaning (strict SI)**
-	(No units specified)
1/T	s^{-1} (Hz)
A	rad
A/P	rad/W
A/PT	rad/Ws
A/PTT	rad/Ws^2
A/T	rad/s
A/TT	rad/s^2
F	N
F/L	N/m
F/LL	N/m^2
FL	Nm
FL/A	Nm/rad
FL/L	Nm/m
FLL	Nm^2
FLT/A	Nms/rad
FLTT/AA	Nms^2/rad^2
I	A
L	m
L/T	m/s
L/TT	m/s^2
LLL	m^3
LLL/A	m^3/rad
M	kg
M/L	kg/m
M/LLL	kg/m^3
M/LT	kg/ms
MLL	kgm^2
N	(No units specified)
P	W
PT	J
Q	VAr
T	s
V	V
VI	VA

Table A.2: Allowed Units

APPENDIX B Example External Controller Code In Selected Languages

To assist the user to get started with the coding required for external controllers, this appendix presents a few simple examples.

B.1 Simple example of DLL code written in C

```
#include <stdio.h>
#include <string.h>
#define NINT(a) ((a) >= 0.0 ? (int)((a)+0.5) : (int)((a)-0.5))

extern "C"                          //avoid mangled names
{ void __declspec(dllexport) __cdecl DISCON(float *avrSwap, int *aviFail,
       char *accInfile, char *avcOutname, char *avcMsg);
}

//Main DLL routine
void __declspec(dllexport) __cdecl DISCON(float *avrSwap, int *aviFail,
       char *accInfile, char *avcOutname, char *avcMsg)
{
       char Message[257], InFile[257], OutName[1025];
       float rTime, rMeasuredSpeed, rMeasuredPitch;
       int iStatus, iFirstLog;
       static float rPitchDemand;

       //Take local copies of strings
       memcpy(InFile,accInfile, NINT(avrSwap[49]));
       InFile[NINT(avrSwap[49])+1] = '\0';
       memcpy(OutName,avcOutname, NINT(avrSwap[50]));
       OutName[NINT(avrSwap[50])+1] = '\0';

       //Set message to blank
       memset(Message,' ',257);

       //Set constants
       SetParams(avrSwap);

       //Load variables from Bladed (See Appendix A)
       iStatus = NINT (avrSwap[0]);
       rTime = avrSwap[1];
       rMeasuredPitch = avrSwap[3];
       rMeasuredSpeed = avrSwap[19];

       //Read any External Controller Parameters specified in the User Interface
       if (iStatus == 0)
       {
              *aviFail = ReadData(InFile, Message);  //User to supply this routine
              rPitchDemand = rMeasuredPitch;         //Initialise
       }

       //Set return values using previous demand if a sample delay is required
       avrSwap[44] = rPitchDemand;

       //Main calculation     //User to supply calcs routine
       if (iStatus >= 0 && *aviFail >= 0)
              *aviFail = calcs(iStatus, rMeasuredSpeed, rMeasuredPitch,
                                 &rPitchDemand, OutName, Message);

       //Logging output - example
       avrSwap[64] = 2;                                     //No of outputs
       iFirstLog = NINT(avrSwap[62])-1;                     //Address of first output
       strcpy(OutName, "Speed:A/T;Pitch:A");                //Names and units
       avrSwap[iFirstLog] = rMeasuredSpeed;                 //First Value
       avrSwap[iFirstLog+1] = rMeasuredPitch;               //Second value

       //Return strings
       memcpy(avcOutname,OutName, NINT(avrSwap[63]));
       memcpy(avcMsg,Message,MIN(256,NINT(avrSwap[48])));

       return;
}
```

B.2 Simple example of DLL code written in FORTRAN 90

```
SUBROUTINE DISCON (avrSWAP, aviFAIL, accINFILE, avcOUTNAME, avcMSG)
IMPLICIT NONE

!Compiler specific: Tell the complier that this routine is the entry point for the DLL

!The next two lines are for the case of the Digital Visual Fortran compiler
CDEC$ ATTRIBUTES DLLEXPORT :: DISCON
CDEC$ ATTRIBUTES ALIAS:'DISCON' :: DISCON
!The Lahey LF90 compiler needs this line instead:
DLL_EXPORT DISCON
!For other compilers: read the documentation to find out how to do this

REAL AV_ avrSWAP(*)
INTEGER*1 accINFILE(*), avcOUTNAME(*), avcMSG(*)
INTEGER aviFAIL

INTEGER*1 iInFile(256), iOutName(1024), iMessage(256)
CHARACTER cInFile*256, cOutName*1024, cMessage*256
EQUIVALENCE (iInFile, cInFile), (iOutName, cOutName), (iMessage, cMessage)
INTEGER I, iStatus
REAL rTime, rMeasuredPitch, rMeasuredSpeed, rPitchDemand
SAVE rPitchDemand

!This just converts byte arrays to character strings, for convenience
DO I = 1,NINT(avrSWAP(50))
 iInFile(I) = accINFILE(I)     !Sets cInfile by EQUIVALENCE
ENDDO
DO I = 1,NINT(avrSWAP(51))
 iOutName(I) = avcOUTNAME(I)   !Sets cOutName by EQUIVALENCE
ENDDO

!Load variables from Bladed (See Appendix A)
iStatus = NINT(avrSwap(1))
rTime = avrSwap(2)
rMeasuredPitch = avrSwap(4)
rMeasuredSpeed = avrSwap(20)

!Read any External Controller Parameters specified in the User Interface
IF (iStatus .EQ. 0) THEN
  aviFail = ReadData(cInFile, cMessage)  !User to suppply this routine
  rPitchDemand = rMeasuredPitch          !Initialise
ENDIF

!Set return values using previous demand if a sample delay is required
avrSwap(45) = rPitchDemand

!Main calculation (User to suppply calcs routine)
IF (iStatus .GE. 0 .AND. aviFail .GE. 0) THEN
  aviFail = calcs(iStatus, rMeasuredSpeed, rMeasuredPitch, &
                rPitchDemand, cOutName, cMessage)
ENDIF

!Return strings
DO I = 1,NINT(avrSwap(64))
 avcOutname(I) = iOutName(I)          !same as cOutName(I) by EQUIVALENCE
ENDDO
DO I = 1,MIN(256,NINT(avrSwap(49)))
 avcMsg(I) = iMessage(I)              !same as cMessage(I) by EQUIVALENCE
ENDDO

RETURN
END
```

B.3 Simple example of EXE code written in FORTRAN 90

```
IMPLICIT NONE
LOGICAL lOK
INTEGER iERROR, iUNIT, iFail, iSTATUS, iStarted
REAL rTime, rPitchDemand, rMeasuredPitch, rMeasuredSpeed

!First open the swap file
L_UNIT = 99
OPEN(L_UNIT, FILE='DISCON.SWP', ACCESS='DIRECT', FORM='UNFORMATTED', RECL=4, &
     ACTION='READWRITE,DENYNONE', IOSTAT=iERROR)
IF (iERROR.NE.0) STOP 'Could not open swap file'

!Set initialisation flag
iStarted = 0

!Write zero to record 1
WRITE(iUNIT, REC=1, IOSTAT=iERROR) 0
CLOSE(iUNIT)
IF (iERROR.NE.0) STOP 'Could not write to swap file'

!Wait for Bladed
lOK = .TRUE.
DO WHILE (lOK)
  OPEN(iUNIT, FILE='DISCON.SWP', ACCESS='DIRECT', FORM='UNFORMATTED', RECL=4, &
       ACTION='READWRITE,DENYNONE', IOSTAT=iERROR)
  IF (iERROR.NE.0) STOP 'Could not re-open swap file'
  READ(iUNIT, REC=1, IOSTAT=iERROR) iSTATUS
  IF (iERROR.NE.0) STOP 'Could not read status from swap file'
  IF (iSTATUS.EQ.-1) THEN
    !End of simulation
    lOK = .FALSE.
  ELSEIF (iSTATUS.EQ.0) THEN
    !Still waiting
    CALL SLEEPQQ(1) !Wait 1 millisecond; Compiler-dependent subroutine.  It may be
                    !unneccessary, but may help to prevent problems on a slow network.
  ELSEIF (iSTATUS.EQ.1) THEN
    !Read from swap file
    READ(iUNIT, REC=2, IOSTAT=iERROR) rTime
    READ(iUNIT, REC=4, IOSTAT=iERROR) rMeasuredPitch
    READ(iUNIT, REC=20, IOSTAT=iERROR) rMeasuredSpeed

    IF (iStarted .EQ. 0) THEN
      iFail = ReadData('DISCON.IN')  !User to suppply this routine
      rPitchDemand = rMeasuredPitch  !Initialise
    ENDIF

    !Set return values using previous demand if a sample delay is required
    WRITE(iUNIT, REC=45, IOSTAT=iERROR) rPitchDemand

    !Main calculation (User to suppply calcs routine)
    IF (iStarted .GE. 0 .AND. iFail .GE. 0) THEN
      iFail = calcs(iStarted, rMeasuredSpeed, rMeasuredPitch, rPitchDemand)
    ENDIF

    iStarted = 1

  ELSE
    STOP 'Handshake status incorrect'
  ENDIF

  CLOSE(iUNIT)

ENDDO

STOP
END
```

APPENDIX C The *Bladed* Demo

Bladed can be run for demonstration purposes if no security device is available.

A demo project, **demo_a.prj**, is supplied with ***Bladed*** and will be found in the installation directory. This contains simplified details of a representative but artificial 3-bladed 2MW offshore variable speed pitch regulated turbine. This can be used to run any of the available calculations.

The ***Bladed*** demo is restricted as follows

Operation	**Restrictions**
Modal analysis calculation	Turbine details may not be changed
Turbulent wind generation	Turbulence details may not be changed
Steady calculations and simulations	Turbine details may not be changed; other calculation details may be changed
Post processing calculations	Calculation details and input data may not be changed
Saving project files	Not allowed
Writing reports	Not allowed, but a sample report may be viewed.
Graphics: viewing calculation results	Unrestricted

When running a calculation, a message box warns that the appropriate demo data will be used for the calculation.

APPENDIX D *Bladed Educational*

Bladed Educational is available solely for academic purposes. Certain features of the full code are not available in this version.

The following restrictions apply to the educational version:

Turbine:

- The number of blade radial stations is limited to 10 maximum;
- The number of tower stations is limited to 5 maximum;
- The pitch actuator model is fixed to a first order lag response with 0.3s time constant. No other pitch actuator models are allowed.
- The external controller facility is not available.

Environment:

- Simulation periods are limited to 60s maximum.
- The "seed" used to initialise the random number generators used to synthesise wind turbulence and wave time series is fixed.
- Only the longitudinal component of wind turbulence can be used in simulations. (The other components can be synthesised, but will be ignored in simulations.)

Calculations and post-processing:

- Model linearisation is not available, except as a demo calculation.
- The following post-processing calculations are not available, except as demo calculations:
  - ultimate loads
  - ultimate load cases
  - linear model

REFERENCES

[1] Germanischer Lloyd, Rules and Regulations IV, Non-marine technology, Part 1 – Wind Energy, Section 2.1993.

[2] MATLAB – the Language of Technical Computing, www.mathworks.com.