

BLADED EDUCATIONAL INTRODUCTORY GUIDE

# BLADED

Version: 1.1  
Date: October 2014  
DNV GL - Energy



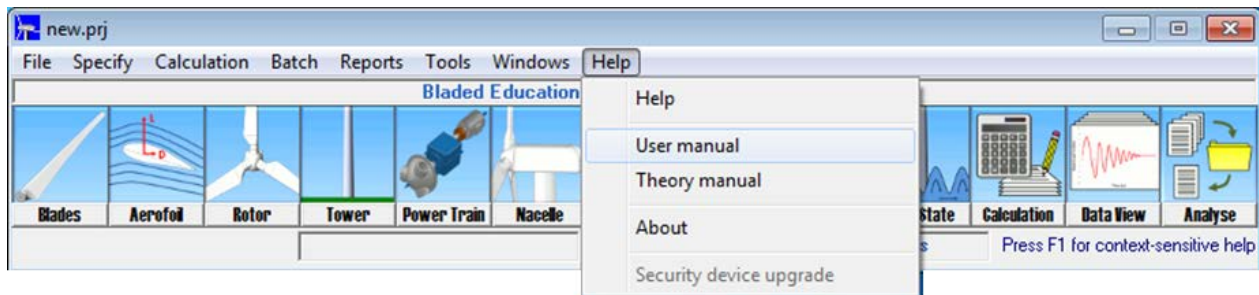
# About this Guide

Welcome to Bladed!

This guide provides a brief introduction to the Educational Version of Bladed. It contains descriptions of the input modules, the most common calculations and a selection of questions for investigation. A worked example of one of the questions is also included.

## For more information:

For more detailed explanations of any of the features of Bladed, or of the theory behind the calculations, a comprehensive User Manual, and Theory Manual are included within Bladed. These can be selected from the Help menu in the user interface, as shown below:



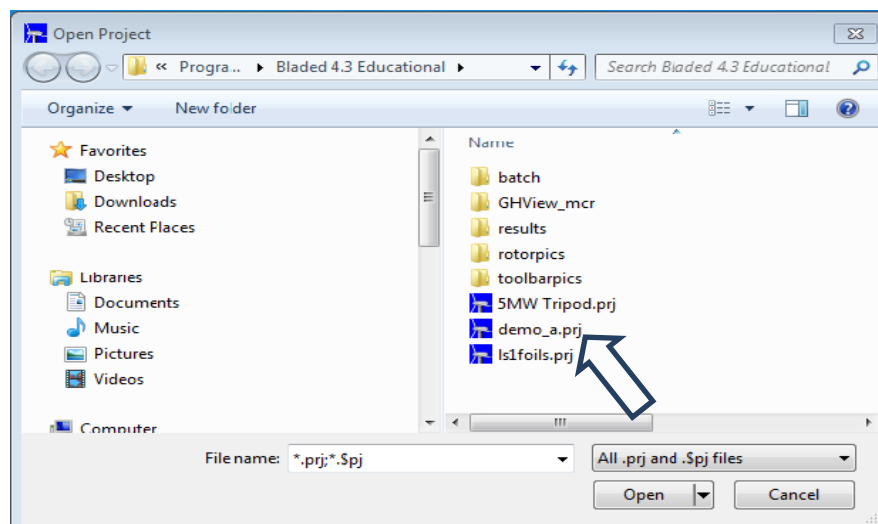
## Quick Help shortcut:

Simply pressing the 'F1' key at any time is a handy shortcut to the User Manual. This will automatically open the page of the Manual that is relevant to the part of Bladed you are working with.

## Demo Turbine:

The Educational version of Bladed comes complete with a demo turbine that has all of the parameters pre-loaded. Opening this Bladed model and exploring its features is a great way to familiarise yourself with the software.

To open the demo turbine go to 'File', 'Open Project' and select demo\_a.prj as shown below:



Enjoy using Bladed!

Version IG1.1 © DNV GL 2014



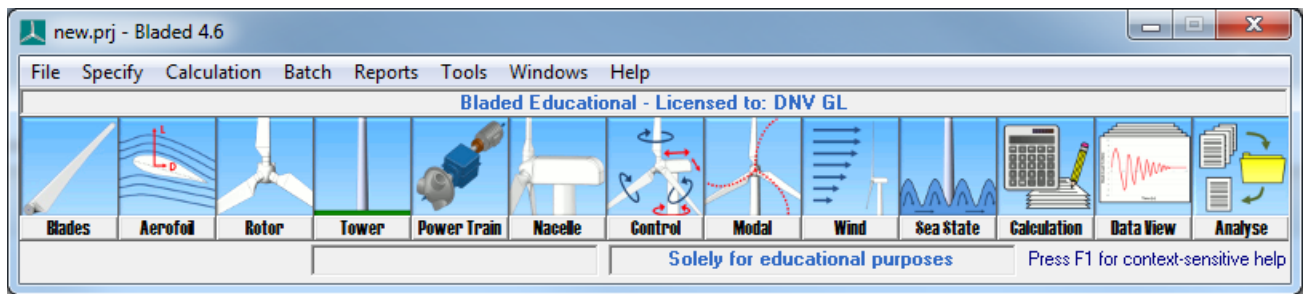
# Contents

<b>INTRODUCTION TO BLADED</b>	<b>1</b>
<b><u>THE INPUTS</u></b>	<b><u>4</u></b>
BLADES	5
AEROFOIL	7
ROTOR	9
TOWER	10
POWER TRAIN	11
NACELLE	13
CONTROL	14
MODAL	15
WIND	16
SEA STATE	17
<b><u>SELECTED MAIN CALCULATIONS</u></b>	<b><u>18</u></b>
CREATING AND USING TURBULENT WIND FILES	19
AERODYNAMIC INFORMATION	21
STEADY POWER CURVE	22
POWER PRODUCTION LOADING	23
<b><u>SELECTED POST PROCESSING CALCULATIONS</u></b>	<b><u>24</u></b>
BASIC STATISTICS	25
AUTO SPECTRUM	26
CHANNEL COMBINATION	27
RAINFLOW CYCLE COUNT	29
<b><u>INVESTIGATION QUESTIONS</u></b>	<b><u>31</u></b>
<b><u>WORKED EXAMPLE</u></b>	<b><u>32</u></b>



# Introduction to Bladed

## Overview:

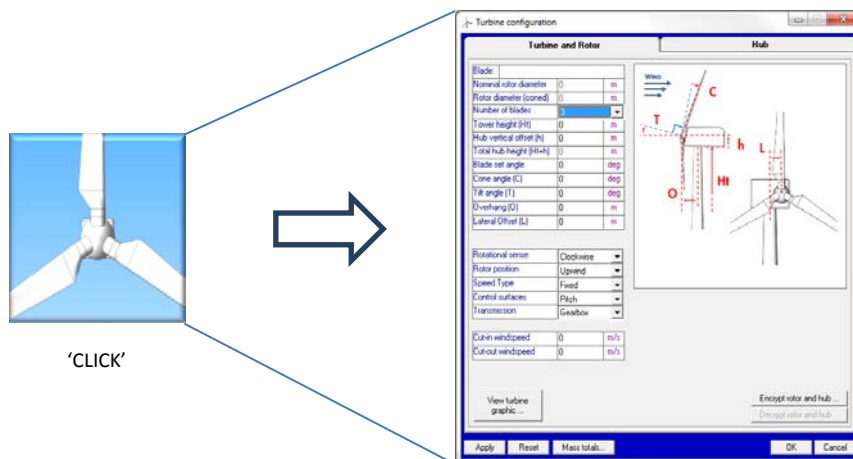


Bladed is a wind turbine design and modelling tool. The user enters parameters to define the turbine, the wind conditions, and the sea state (for offshore turbines). Bladed then calculates structural loads and dynamic responses, plus power and energy values and other data.

## User interface:

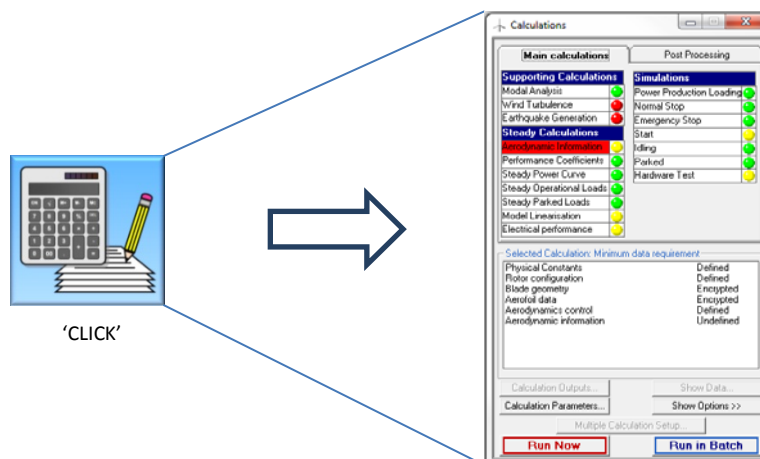
### 1. INPUTS

The first ten icons are inputs. Clicking on each one opens a data entry window. E.g 'Rotor'



### 2. CALCULATIONS

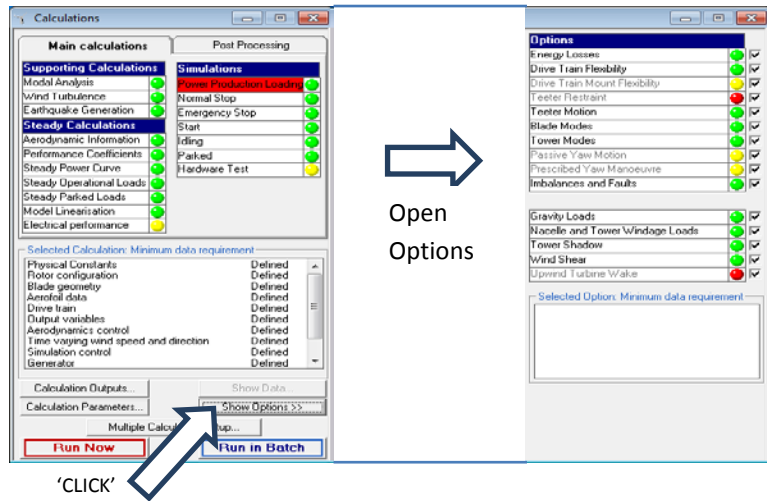
The calculations available are shown in the calculation window. A green light means Bladed has all the information it needs to carry out that calculation. Red or yellow means more data needs to be entered.



When a calculation has the green light the user can select it and click 'Run Now'. Bladed asks where to save the results, and then performs the calculation.

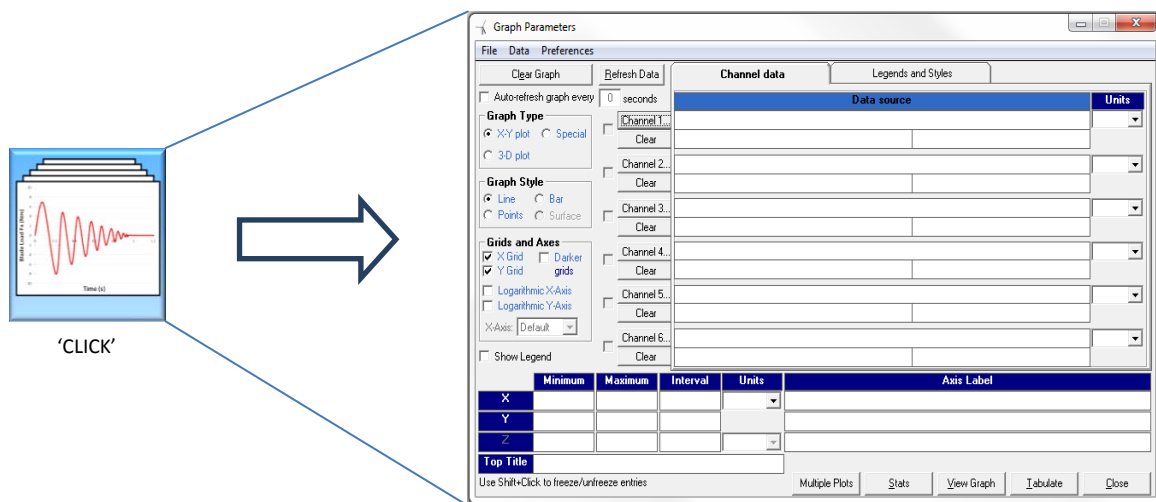
### 3. CALCULATION OPTIONS

Click 'Show Options' in the calculation window. This opens a list of settings that can be enabled or disabled within the model. This is a very useful feature for investigating the effect of specific parameters on power production loading



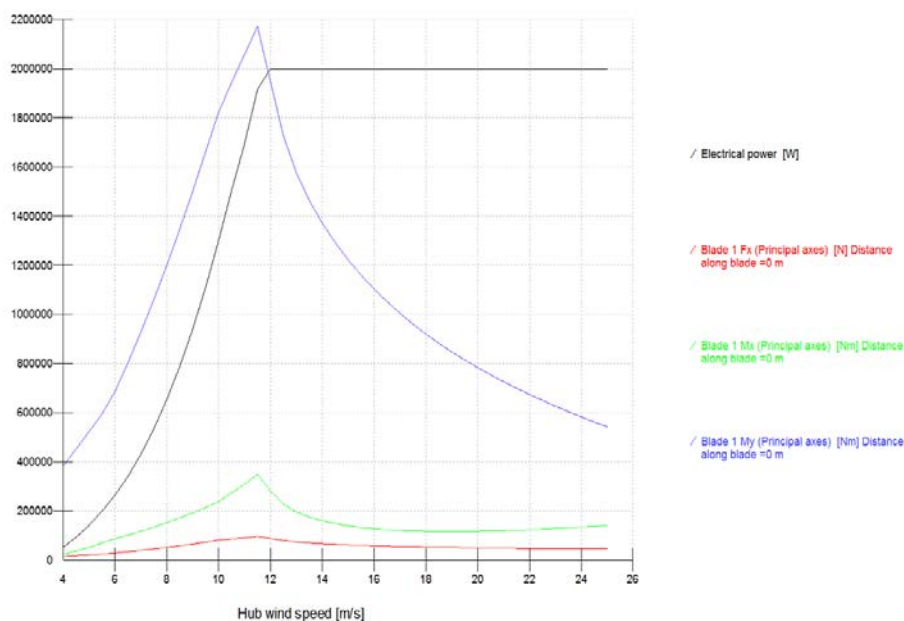
### 4. OUTPUTS

The Data View icon allows calculation results to be displayed. Data can also be exported to Excel.



To produce a graph, click on 'Channel 1'. A window appears where you can choose the data series required. Clicking 'View Graph' at the bottom displays the graph. 'Tabulate' allows data to be exported to Excel.

Up to six data series can be displayed on the same graph using the six available channels. An example graph showing four quantities plotted simultaneously appears below:









**DNV·GL**



# **The Inputs**

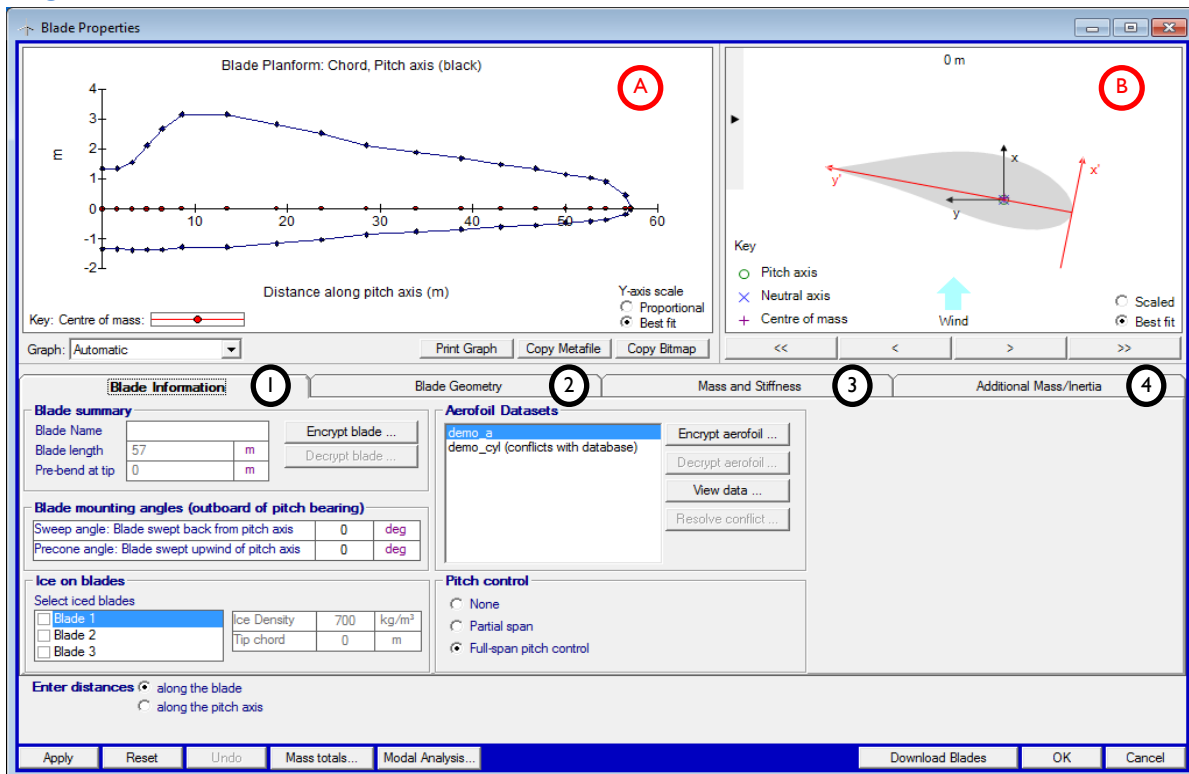
# Blades



## Overview

The shape and design of wind turbine blades has a huge influence on overall turbine performance, power output, and turbine loading. In this part of Bladed, we define the **blade geometry**, **aerofoil sections**, **mass** and **stiffness**, as well as other blade details.

## What you see



A

This shows a schematic of the length of the blade. Various blade parameters can be displayed by selecting from the dropdown menu immediately below the graph.

B

This shows a cross-sectional 'slice' through the blade, which changes as we move from one station to the next in the Blade Geometry tab.

### 1 Blade Information : general information about the blade

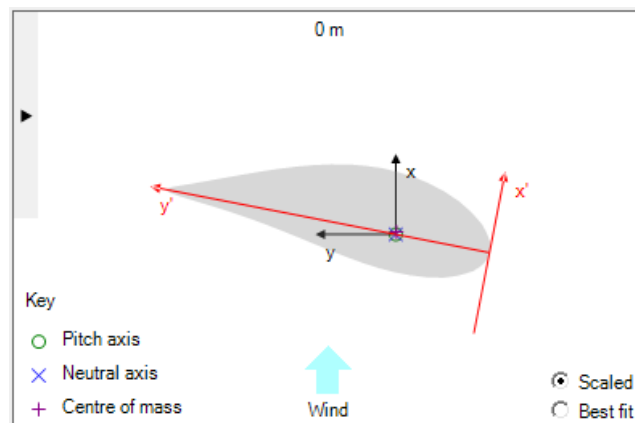
Here we can view a summary of the blades and blade mounting angles, select iced blades, and view the list of aerofoil profiles used in the current blade model.

### 2 Blade Geometry : defining the blade stations

This section allows us to build up the shape of the blade as a series of individual stations, which can be added and deleted using the buttons on the right hand side. As we add a new station, we first define the distance from the root of the blade. We can then edit the various blade characteristics. (Note that the aerofoil datasets need to be entered first through the Aerofoil window as explained in the next section).

Moving the mouse over a particular characteristic displays a precise definition of that parameter.

A worthwhile exercise would be to vary the values of each of these parameters in turn for one particular blade station, and see how this affects the cross section image in the top right hand corner.



### 3 Mass and Stiffness

Here we detail the distribution of mass on the blade, as well as other characteristics such as stiffness. The check boxes on the left allow characteristics to be enabled or disabled depending on the modelling complexity required. Moving the mouse over a particular characteristic displays a precise definition of that parameter.

### 4 Additional Mass/Inertia

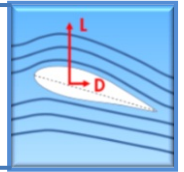
In this section we define the location and mass of any added masses on the blade in a similar fashion to the Blade Geometry, stating the **distance along the blade**, the **mass**, and the **chordwise position in  $x'$  and  $y'$** . We can also give details of **vibration dampers** within the blade, and any **additional pitching inertia**.

From this toolbar we can also select 'Mass totals...' to see values for the **blade mass**, **moments of mass**, and **moment of inertia**.

### Completing the blades

Once the blade design has been finished, click 'OK' in the bottom right hand corner.

# Aerofoil



## Overview

To model the aerodynamics of turbine blades, Bladed uses the **lift**, **drag** and **pitching moment** characteristics of the blade defined at each station to calculate the resulting aerodynamic forces and moment along the blades. The data for these characteristics is known as an **aerofoil dataset**. A number of these datasets can be combined to define an **aerofoil section**. This screen allows the information in these datasets to be entered, observed and edited. Once entered, these datasets will then be available for use in the Blade Geometry window.

## What you see

Initially the Aerofoil Database screen will appear blank with no data displayed.

There are several ways to enter data. Firstly, data can be loaded directly from Bladed. Click 'Load' at the bottom left for a list of available data, and choose demo\_a if using the demo turbine. Secondly, 'Import' allows an existing aerofoil dataset to be uploaded. Alternatively, data can be pasted directly from a textfile, or from Excel. Finally, data can be entered manually or edited, using the input fields at the bottom right.

Once entered, aerofoil datasets can be saved for future use. 'View Data' generates an instant graph of the aerodynamic characteristics plotted against angle of attack.

$\alpha$	$C_L$	$C_D$	$C_M$
150.00	-0.679	0.481	-0.305
154.00	-0.618	0.352	-0.268
156.00	-0.629	0.303	-0.267
158.00	-0.722	0.290	-0.302
160.00	-0.814	0.276	-0.338
161.00	-0.909	0.261	-0.373
162.00	-1.083	0.233	-0.439
163.00	-1.257	0.205	-0.505
163.40	-1.324	0.188	-0.531
164.00	-1.271	0.180	-0.511
166.00	-1.096	0.132	-0.446
168.00	-0.921	0.084	-0.381
170.00	-0.785	0.062	-0.333
172.00	-0.661	0.048	-0.290
176.00	-0.398	0.029	-0.189
180.00	-0.088	0.035	-0.041

## 1 Aerofoil data

In this section we can name the aerofoil dataset and add a description in the comments section. We can also define or edit general parameters, such as **Thickness to Chord ratio**, **Reynolds Number** and **Pitching Moment Centre**.

## 2 Look-up table

This table contains the aerodynamic characteristics of the dataset. The first column shows the angle of attack, and the subsequent columns show the lift, drag and pitch moment coefficients for each angle of attack. This information can be edited using the buttons just below the table, and new entries can also be added to further define the data set.

### Using aerofoil interpolation

Once the required aerofoil datasets have been completed, it is possible to combine several of these datasets using aerofoil interpolation.

In the **Blades** screen, there is a row in the **Blade Geometry** tab labelled **Foil Section**. If we select 'Define...' from the drop-down list at a particular blade station, we see the following screen.

Define Aerofoil Sections	
Blade Station:	26.4074m
Section Type:	Normal
Section Number:	3
New	
Aerofoil Datasets For Interpolation	
Reynolds Number Interpolation	
Thickness/Chord Ratio	LS1m13r2
Interpolation	LS1m17r2
OK Cancel	

A The drop-down list here can be used to assign a number label to each combination of aerofoil datasets. The appropriate label number is then selected back in the blade geometry tab.

B This table can be used to interpolate between aerofoil datasets with different Reynolds numbers and thickness/chord ratios. In the example above, the Foil Section number 1 corresponds to an aerofoil section that interpolates by thickness/chord ratio between the "LS1m13r2" and "LS1m17r2" datasets.

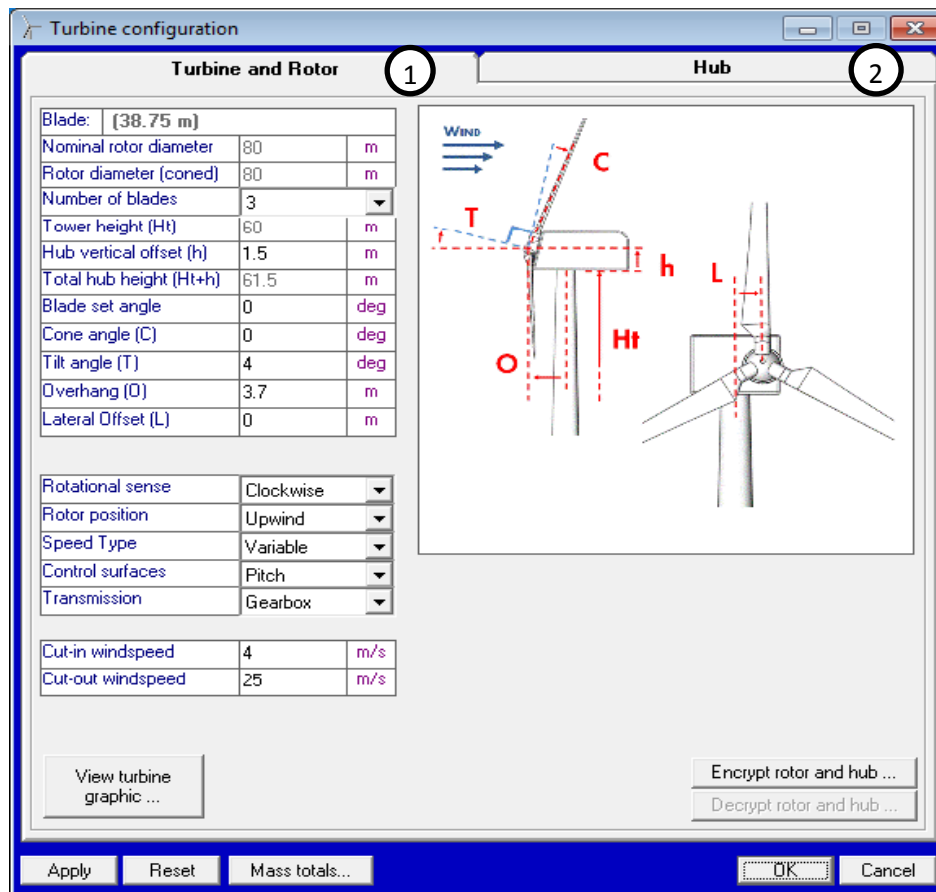
# Rotor



## Overview

The exact design and orientation of the rotor and hub is dependent on many factors. From this screen, we define the parameters and dimensions of the rotor and hub.

## What you see



### 1 Turbine and Rotor

In the first tab we define the dimensions of the turbine rotor, and key information on its operation and design such as **number of blades**, **cone angle** and **overhang**, **speed type** and the turbine's **cut-in and cut-out windspeed**. We can also define the **rotation orientation** of the turbine (clockwise or anticlockwise) and the **position** of the rotor relative to the wind (upwind or downwind).

### 2 Hub

Here we can edit information on the dimensions of the blade root **spinner**; and the **mass** of the various components of the hub.

### 3 Viewing the turbine

Once the blade, tower and rotor have been defined in Bladed, a fully rendered 3D model of the turbine model can be viewed by clicking on '**View turbine graphic...**'. It is possible to zoom in and out, and rotate the model using the icons in the top toolbar.

# Tower



## Overview

In the educational version of Bladed, only **tubular axisymmetric** tower designs can be modelled. In the full version of the software, the multi-member tower option can also be selected.

## What you see: Tubular axisymmetric tower

Select Tubular Axisymmetric in the section labelled **1**. The screen appears as below:

The screenshot shows the 'Tower' software window. It contains several sections: 'Tower properties' with checkboxes for Geomtry, Mass, Stiffness, and Torsional degree of freedom; 'Environment' with options for Land or Sea and associated coefficients; 'Tower Structure' with a table for station properties and buttons for adding/deleting stations; 'Foundations and Ground / Sea-bed Properties' with options for translational and rotational motion; and a 'Graph' section for visualizing the tower. Numbered callouts point to specific features: 1 points to the 'Tower type' dropdown, 2 points to the material table, 3 points to the environment selection, 4 points to the tower structure table, 5 points to the tower geometry graph, and 6 points to the foundation properties section.

Tower Station Number	1	2	3	4	5
Height above surface (m)	-15	15	35	50	60
Outside diameter (m)	3.5	3.19231	2.88462	2.73077	2.6
Mass per unit length (kg/m)	2935.71	2214.09	1492.47	1131.66	824.97
Bending Stiffness (Nm²)	2.46E+11	1.43E+11	7.81E+10	5.65E+10	4.22E+10
Torsional stiffness (Nm²)					
Polar mom. of inertia (kgm)					
Wall thickness (mm)					
Material					
Hydro drag coeff.					
Hydro inertia coeff.					

- 1 Here we are able to enable and disable the modelling of **mass**, **stiffness** and **torsional degree of freedom** in the tower design.
- 2 This is a look up table of the **materials** used, in which the **material name**, **density** and **Young's modulus** are entered.
- 3 **Land** or **sea** environment is selected, and the relevant drag and inertia coefficients are defined.
- 4 In this section we build up the **tower structure** station by station. The parameters which need to be defined depend on which boxes have been checked in section 1. Stations are ordered in ascending height. The material used for each segment can be selected too, providing that it has been entered in section 2. Otherwise, mass and stiffness properties can be entered explicitly for each tower station. Point masses can also be added if required.
- 5 This displays a cross section through the tower structure.
- 6 In this section we can define the **foundations** of the turbine, and properties of the **ground** or **sea bed**. A **vibration damper** can also be applied.

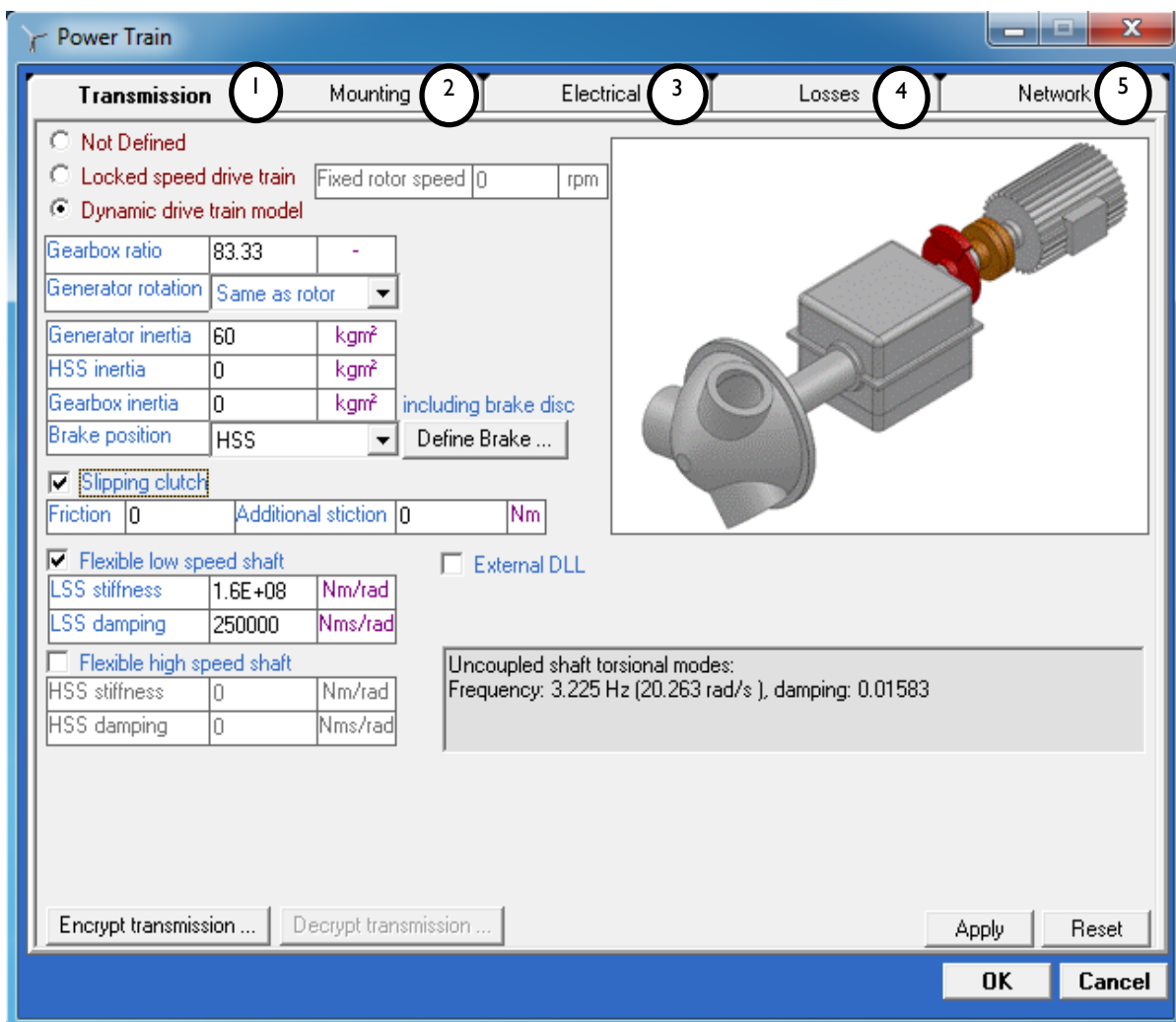
# Power Train



## Overview

Realistic modelling of the power train is essential in determining the **electrical power** that can be generated. The Power Train module of Bladed allows the turbine's **power transmission** and **gearbox** to be defined, along with the **mounting** of the drive train, the **electrical network** and its connectivity.

## What you see





## 1 Transmission

In this section we enter key information on the drive train set up. We first select the drive train model, either **fixed speed** or **dynamic**. We can then supply the **gearbox ratio** and **drive train inertias**, and also add **flexibility** to both the low and high speed shafts.

The image within the window displays a simplified model of the drive train we have defined, showing the **brake position** (red) and the position of a **slipping clutch** (gold). The box at the bottom right shows the torsional modes of the shaft.

## 2 Mounting

In this section we can choose to model **flexibility** of the **drive train pallet** or **gearbox mount**, supplying the **stiffness**, **damping** and **inertia** in each case.

## 3 Electrical

In this tab we provide information on the type of **generator**, and important generator parameters, such as maximum and minimum **torque**, and the **time constant**.

## 4 Losses

This is an important section, detailing the **mechanical** and **electrical losses** in the power train. For the mechanical losses, these can be entered as a function of **torque** or **power** using a look up table. In the electrical losses, this can either be a linear relationship or a look-up table.

## 5 Network

The final tab is where we define the **network** that our turbine is connected to, giving the number of additional turbines and the **voltage** of the network.

# Nacelle



## Overview

A straightforward data entry module in which the dimensions, mass distribution and inertia of the nacelle are defined.

## What you see

A screenshot of a software window titled "Nacelle". The window has a standard Windows-style title bar with minimize, maximize, and close buttons. The main content area is divided into two sections. The first section, "Assign Geometry", is checked and includes a "Nacelle windage" option with "Drag only" selected. It contains a table for geometric parameters. The second section, "Assign Mass (excluding rotor and hub)", is also checked and contains a table for mass and inertia parameters. At the bottom, there are buttons for "Encrypt nacelle ...", "Decrypt nacelle ...", "OK", and "Cancel".

Assign Geometry		
Nacelle windage: <input checked="" type="radio"/> Drag only <input type="radio"/> As an aerofoil		
Nacelle length	m	6
Nacelle height	m	2.5
Nacelle width	m	2.5
Nacelle drag coefficient	-	1.6
Location of front of nacelle measured from tower axis	m	7.2
Aerofoil dataset (from database)		

Assign Mass (excluding rotor and hub)		
Mass	kg	72000
Position of Centre of Mass to side of tower axis	m	0
Height of Centre of Mass, relative to tower top	m	1.3
Position of Centre of Mass in front of tower axis	m	-0.6
Yaw inertia (about tower axis)	kg m <sup>2</sup>	215000
Nodding inertia (about centre of mass)	kg m <sup>2</sup>	0
Rolling inertia (about centre of mass)	kg m <sup>2</sup>	0

# Control



## Overview

The controller is often described as the brains of a turbine. A good controller ensures that the turbine operates at maximum efficiency, and also reduces structural loading by monitoring operation and responding dynamically to avoid resonances. The controller also protects the turbine by triggering shutdowns in the event of faults, grid failures or extreme climatic conditions.

## What you see

A screenshot of a software window titled 'Control Systems'. It contains several tabs and sections. The 'Power Production Control' tab is active, showing options for 'Stall Regulated', 'Fixed Speed', and 'Variable Speed'. The 'Supervisory Control' tab is also visible, showing buttons for 'Start...', 'Normal Stop...', 'Emergency Stop...', 'Brakes...', 'Yaw Control', 'External Controller...', 'Parked...', 'Idling...', and 'Pitch Actuator...'. The 'Torque-speed curve below rated' section has a 'Calculate...' button and a table with parameters like 'Optimal mode gain', 'Minimum Generator Speed', and 'Optimal Mode Maximum Speed'. The 'Above Rated: Pitch Regulated Speed' section has a table with parameters like 'Minimum Pitch Angle', 'Maximum Pitch Angle', 'Demanded Generator Torque', and 'Demanded Generator Speed'. The 'Transducers' section has input fields for 'Power Transducer time constant' and 'Speed Sensor Time Constant'. The 'Pitch control: software rate limits' section has input fields for 'Minimum pitch rate' and 'Maximum pitch rate'. The 'Torque Control' section has buttons for 'Define...' and 'Discrete External Controller'. The 'Pitch Control' section has buttons for 'Define...' and 'Discrete External Controller'. The window has a status bar at the bottom with buttons for 'Safety System...', 'Turbine faults...', 'Apply', 'Reset', 'OK', and 'Cancel'. There are red circles with numbers 1 and 2, and letters A and B, highlighting specific areas in the window.

Controllers are complex, and unsurprisingly the control tab displays a large range of parameters. However, in practice, a controller is often provided, meaning it's not necessary to define all of these variables each time.

### 1 Power Production Control

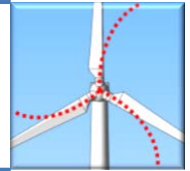
The overall control strategy is defined here. The simplest choice is fixed speed and fixed pitch (stall regulated). Try selecting it – you will see that virtually no control is necessary because everything is held constant.

The standard turbine control strategy is variable speed operation up to rated windspeed (controlled by varying the torque of the generator – [section A](#)), and variable pitch operation above rated windspeed (changing the pitch of the blades - [section B](#)). With this selected, the window appears as above, with many options.

### 2 Supervisory Control

These are a set of control options that define a turbine's response to a range of scenarios. Often these are safety features, designed to protect the turbine from damage.

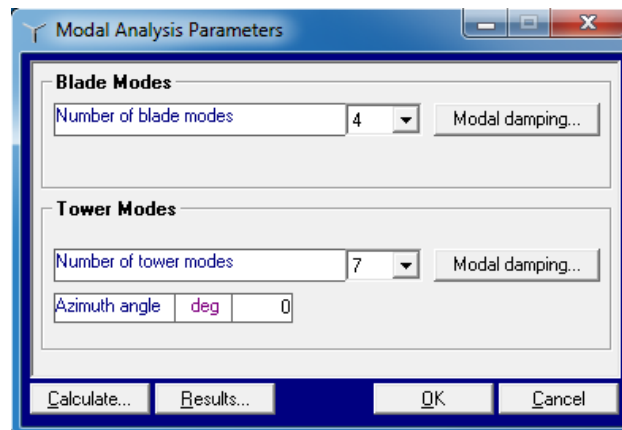
# Modal



## Overview

This module calculates the frequencies and mode shapes of a specified number of modes. The steady deflections are important for steady loads, while the vibrational frequencies are used in simulations. Modal Analysis needs to be run after a model has been completed, and before any Bladed dynamic simulations are run.

## What you see



1. First, select the required number of modes, then click 'Modal damping' and enter the damping values for each mode. These values are often provided by the turbine manufacturer. If not, a default value of 0.5% (0.005) can be used.
2. Click 'Calculate'. The calculation takes just a few seconds to run.
3. Clicking 'Results' displays the modal frequencies in the following screen:

Frequencies (Hz)	Damping ratio	Mode type
0.958	0.01	Blade: Flapwise normal mode
1.458	0.01	Blade: Edgewise normal mode
2.860	0.01	Blade: Flapwise normal mode
4.840	0.01	Blade: Edgewise normal mode
0.470	0.005	Tower side-side translational attachment mode
0.472	0.005	Tower fore-aft translational attachment mode
3.098	0.005	Tower side-side rotational attachment mode
3.629	0.005	Tower fore-aft rotational attachment mode
4.889	0.005	Tower side-side normal mode
4.889	0.005	Tower fore-aft normal mode

View Mode Shape...

Close

4. Select a mode and click **View mode shape** for a graphical display of the modal deflections. The mode shape can be animated by clicking on the double arrow icon on the graphics window toolbar.

**TIP:** 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. Typically, 4 blade modes are calculated, and 7 tower modes.

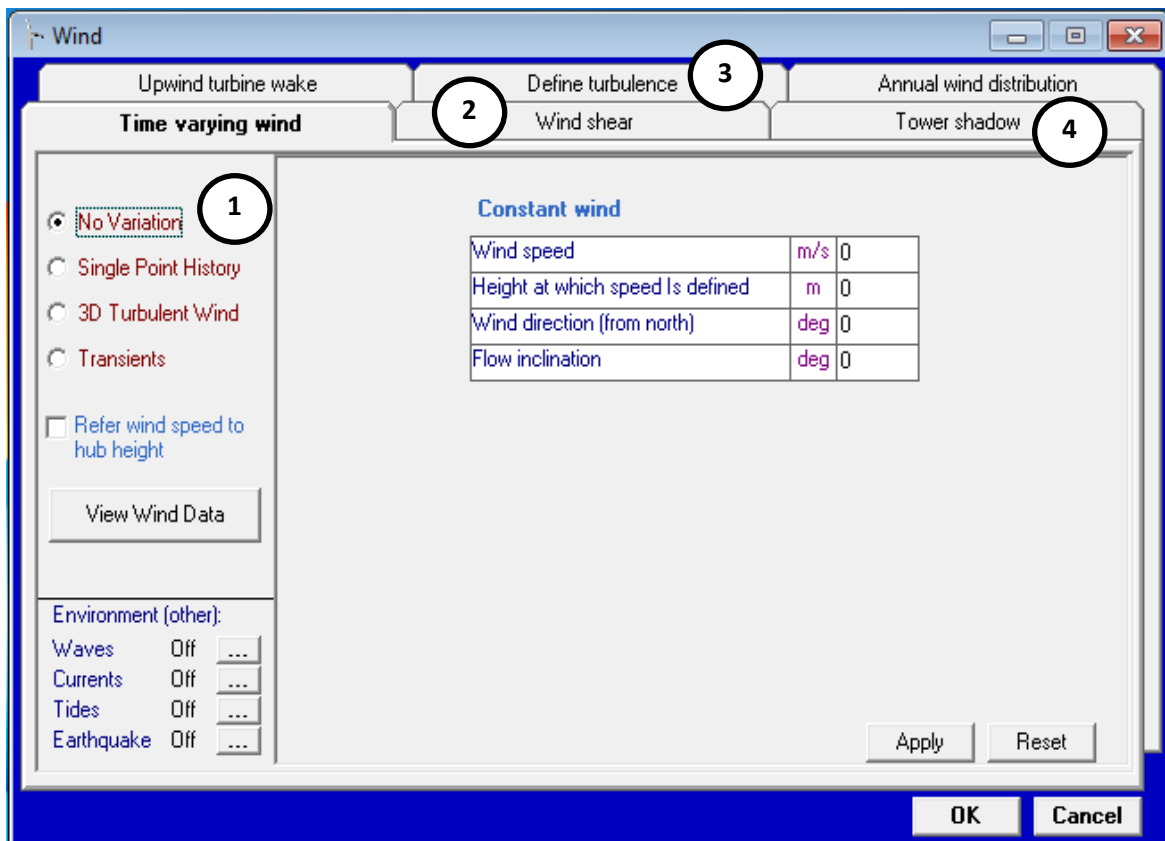
# Wind



## Overview

Defining the incident wind on the turbine is very important. This module allows any wind profile to be simulated, from a constant, steady flow to a full 3D turbulent wind field. The data entered here is not part of the model itself, and will only need to be entered before running a particular simulation.

## What you see



To get started, the four tabs highlighted above are probably the most important:

- 1 Time varying wind**

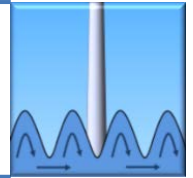
The appearance of this tab varies depending on which of the four red options at the top left is selected. 'No Variation' produces the simplest wind profile, while '3D Turbulent Wind' is used for more realistic simulations. 'Transients' allows isolated wind gusts to be simulated.
- 2 Wind Shear**

This straightforward tab allows the wind shear profile to be defined.
- 3 Define Turbulence**

For many simulations, the first step is to generate a turbulent wind field. This is then saved, and the file location selected on the 'Time varying wind' tab. See Bladed user guide (press F1), or page 19 of this guide, for a detailed explanation.
- 4 Tower Shadow**

This defines the distortion to the steady-state mean wind field due to the presence of the tower. Several options are available. Press F1 help for details.

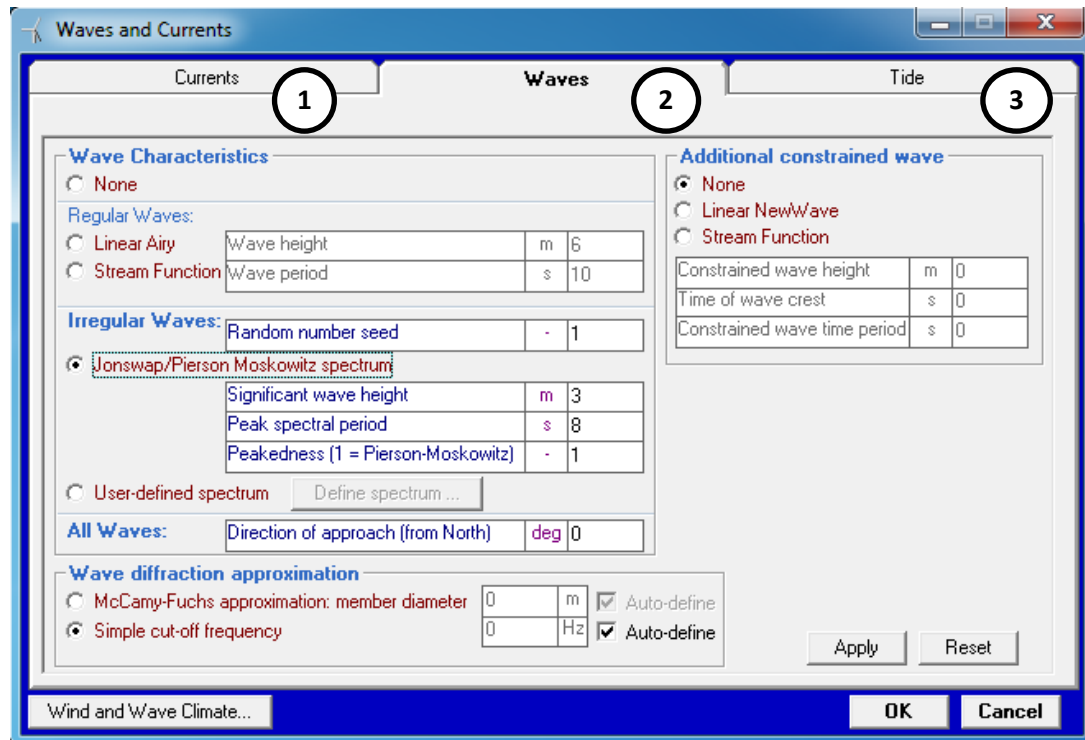
# Sea State



## Overview

This module is used for offshore turbines where currents, waves and tides provide extra sources of loading on the structure.

## What you see



### 1 Currents

This tab allows sea currents to be defined. Near-surface, sub-surface or near-shore currents can be defined. Press F1 Help for a clear description of these possible current components.

### 2 Waves

Waves can be an important source of structural loading. Regular or irregular waves can be simulated. Diffraction around structural members can alter the energy transfer dynamics between the waves and the structure. Two options for modelling this effect are offered.

### 3 Tide

This tab allows the tide height above mean sea level to be entered.

The 'Wind and Wave Climate' button at the bottom links to the 'Wind' module window.



DNV·GL



## **Selected Main Calculations**

## Creating and using Turbulent Wind Files

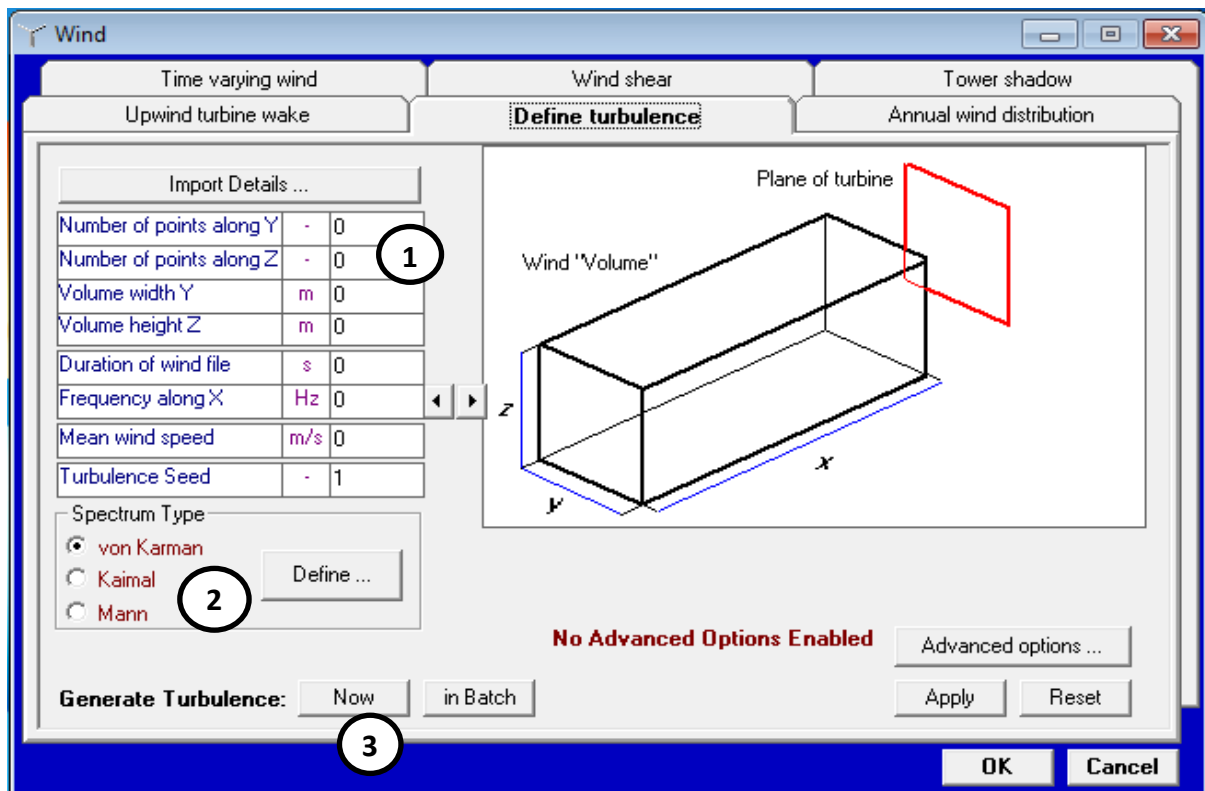
Supporting Calculations	
Modal Analysis	
Wind Turbulence	
Earthquake Generation	

### Overview

When testing a turbine design, it is important to subject the turbine to the kind of turbulent wind conditions that it will face once constructed. To do this in Bladed we create a **turbulent wind file**: a time history of the variation and distribution of the wind within a volume into which our turbine will sit, almost like a virtual wind tunnel. The information for this turbulent wind will then be saved in a file, which can be used in future simulations. In this section we explain how this is done.

### Creating the wind file

To perform this calculation, go to the **Wind** module, and select the '**Define turbulence**' tab. This window can also be opened by selecting 'Wind Turbulence' from the Calculation menu.



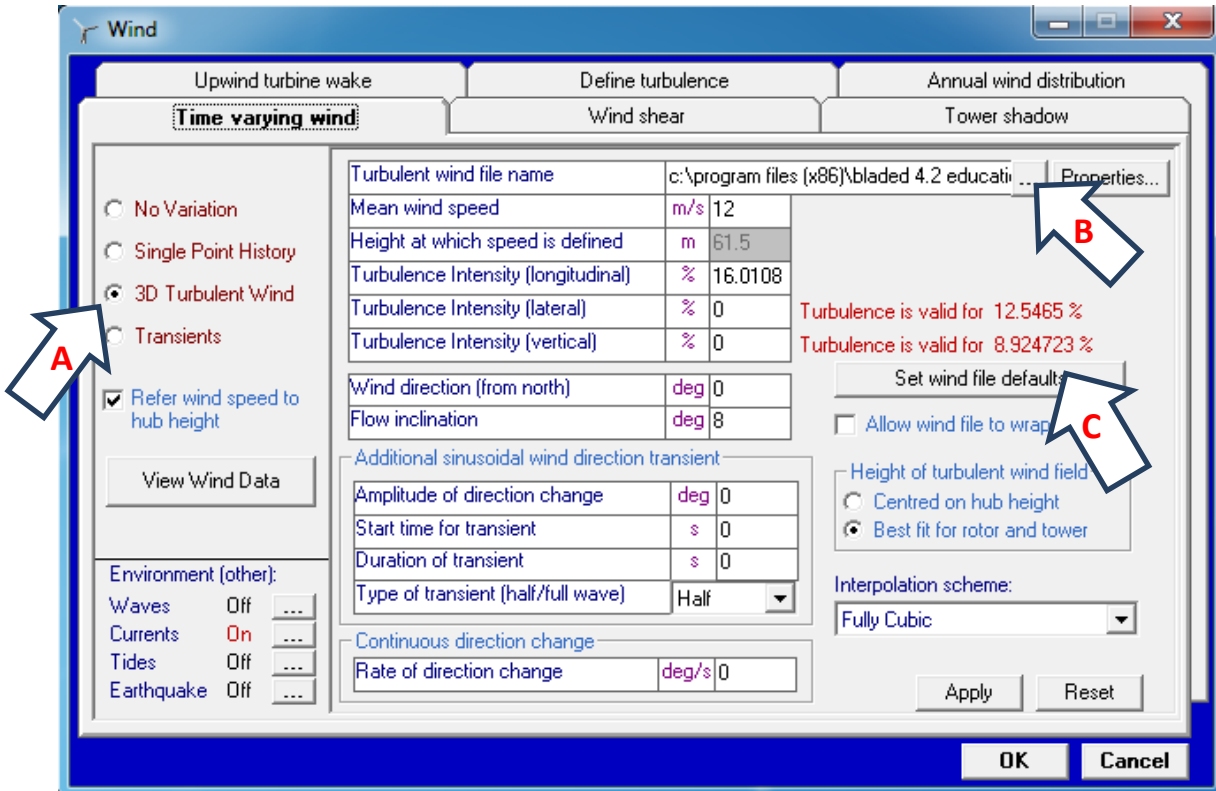
1 Here we enter the values and parameters which will determine the characteristics of our turbulent wind field.

The dimensions X and Y of this field (shown as the black container in the right hand image) should importantly be larger than the main dimensions of the turbine, with the height Z ideally being greater than the total height of the turbine, and Y being greater than the rotor diameter. We also need to define the duration of the wind file and the mean wind speed. The number of points should be chosen so that points are spaced no more than 5 or 6m apart.



- 2 Several different turbulence models can be used. They can be chosen and modified here.
- 3 Once all the necessary details have been entered, we can go on to generate the wind file. This should only take a short time, depending on the duration and size of the wind field, and Bladed will ask for a file location to store the wind file.

## Using the wind file



Once the wind file has been generated, we can use it in a calculation by configuring the simulation for turbulent wind.

Go to the **"Time varying wind"** tab and select **'3D Turbulent Wind'** on the left hand side (A).





In the top dialog box 'Turbulent Wind File Name', select the wind file generated previously (B).

Click 'Set wind file defaults' (C) to set the mean wind speed value. If the von Karman turbulence model has been used to create the turbulent wind file, this will also generate the longitudinal, lateral and vertical turbulence intensity values. For other turbulence models, the turbulence intensity values must be entered manually.

The turbulent wind field is then ready to be used.

**NOTE:** In the Educational Version of Bladed, only the longitudinal component of turbulence is used in simulations.

# Aerodynamic Information

Steady Calculations	
Aerodynamic Information	
Performance Coefficients	
Steady Power Curve	
Steady Operational Limits	

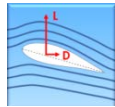
## Overview

The Aerodynamic Information calculation provides aerodynamic characteristics for the blade, for each individual blade station. This includes the angle of attack, inflow factors and the lift, drag and pitching moment coefficients.

## Required components



Blades



Aerofoil



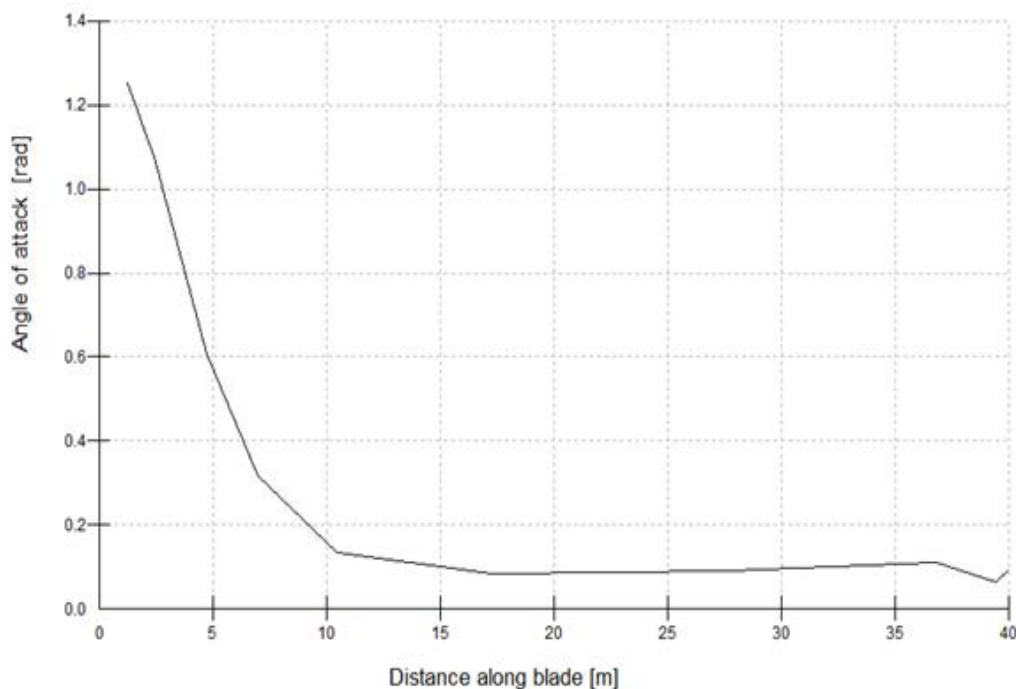
Rotor

## Instructions

1. Complete the information for the modules shown to the left
2. Click on 'Calculation Parameters' at the bottom of the calculation window. Ensure the 'Aerodynamics Control' and 'Aerodynamic Information' tabs are complete
3. Run calculation and choose an output location

## Output and Applications

An Aerodynamic Information calculation produces a graph of variables such as Reynolds number and angle of attack (**see below**), as a function of distance along the blade. This can be very useful for preliminary blade design and for assessing the blade's aerodynamic performance.



# Steady Power Curve

Steady Calculations	
Aerodynamic Information	
Performance Coefficients	
Steady Power Curve	
Steady Operational Loads	
Steady Parked Loads	

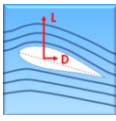
## Overview

A Steady Power Curve calculation determines the turbine power as a function of wind speed, assuming uniform, steady winds with no shear or turbulence. It can also be used to look at how electrical power, torque and thrust vary with wind speed.

## Required components



Blades



Aerofoil



Rotor



Power Train



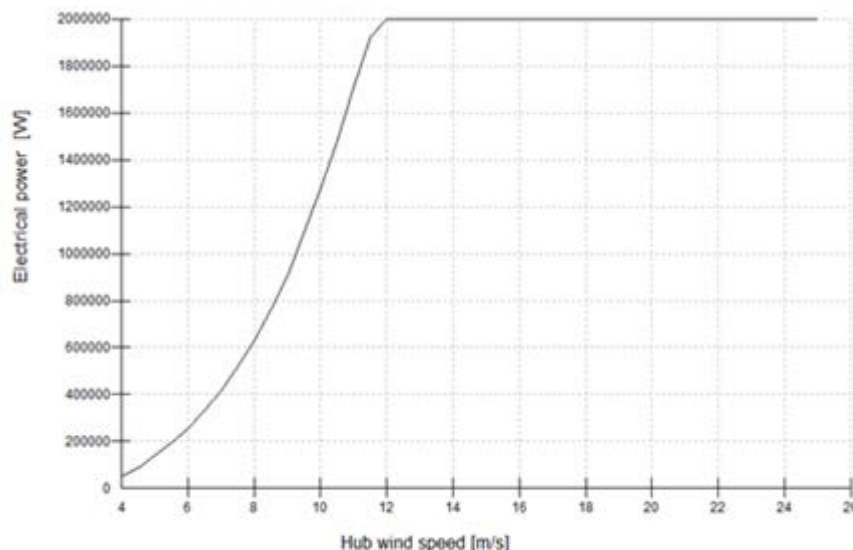
Control

## Instructions





1. Complete the information for the modules shown to the left
2. Click on 'Calculation Parameters' at the bottom of the calculation window. On the 'Power Curve' tab enter the wind speed range and the step size. Ensure the 'Physical Constants' and 'Aerodynamics Control' tabs are complete, as these will also affect the simulation
3. Run calculation and choose an output location

## Output and Applications

This calculation produces a graph displaying the steady power (**see below**), thrust and other variables delivered by the turbine, as a function of steady wind speed between the cut-in and cut-out speeds. This can be used for assessing the turbine productivity over the full range of its operation.



# Power Production Loading

Simulations	
Power Production Loading	
Normal Stop	
Emergency Stop	
Start	

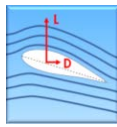
## Overview

A Power Production Loading calculation is a time domain simulation of the wind turbine's response and loading during power production/operation under a range of user defined external conditions.

## Required components



Blades



Aerofoil



Rotor



Tower



Power Train



Nacelle



Control



Modal



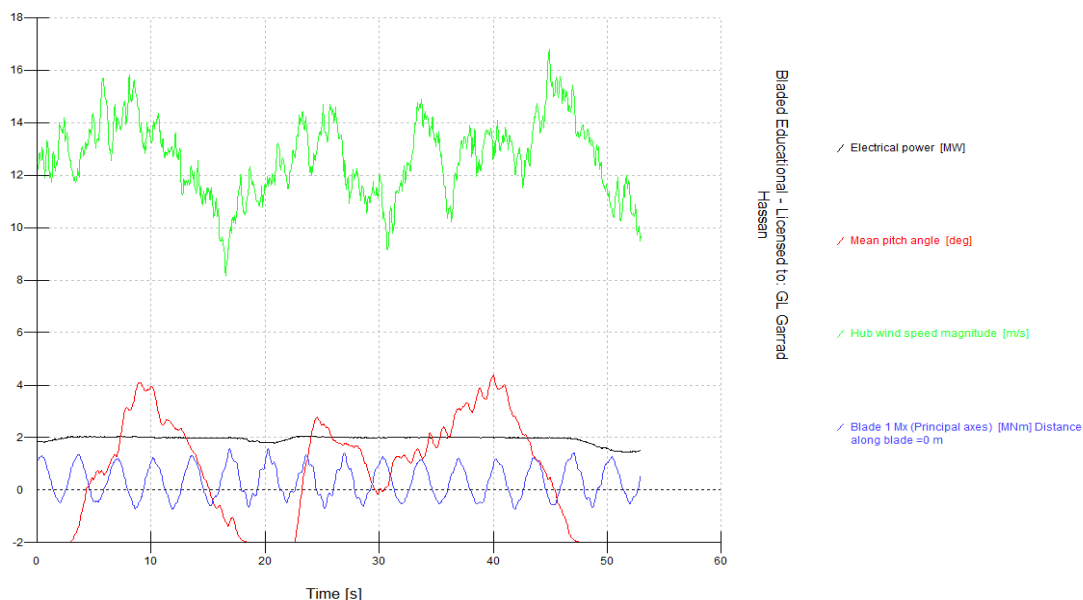
Wind

## Instructions

1. Complete the information for the modules shown to the left
2. Click on 'Calculation Parameters' at the bottom of the calculation window. Ensure the 'Physical Constants', 'Aerodynamics Control' and 'Simulation Control' tabs are complete, as these will also affect the simulation
3. Run calculation and choose an output location

## Output and Applications

A Power Production calculation generates a very wide range of outputs, allowing almost every aspect of turbine performance to be examined. Outputs include wind conditions, power output, rotor speed, pitching angle, controller information, and structural forces and moments (loads). All of these can be displayed graphically or exported to Excel, using the Data View module (icon shown to the right).





DNV·GL



## **Selected Post Processing Calculations**

# Basic Statistics

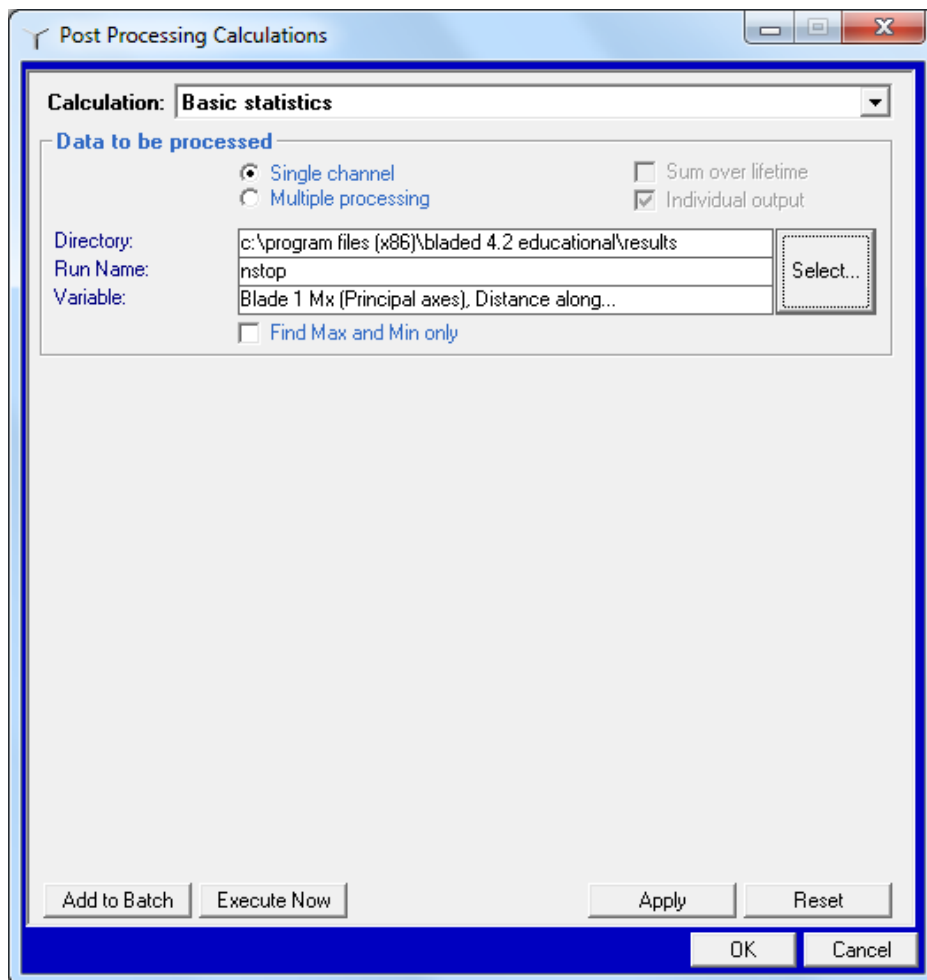


## Overview

The Basic Statistics post-processing calculation provides statistical information on a chosen output signal, including the mean, minimum, maximum, standard deviation, skew and kurtosis.

## Running the calculation

In the 'Post Processing' tab of the Calculations window, click on 'Basic Statistics'. The following window will appear.

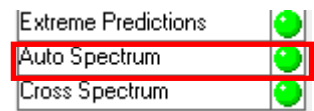


To choose the data for processing, click 'Select'. You may then choose the data for analysis by defining the path, run name and data group. In the Educational version of Bladed only a single channel of data may be selected.

Next click 'Execute now'. You will be asked where to save the results, and the calculation will run.

Click 'Results' to open the Data View window. Define the data channel(s) to view results either graphically or in tabulated form in Excel.

# Auto Spectrum

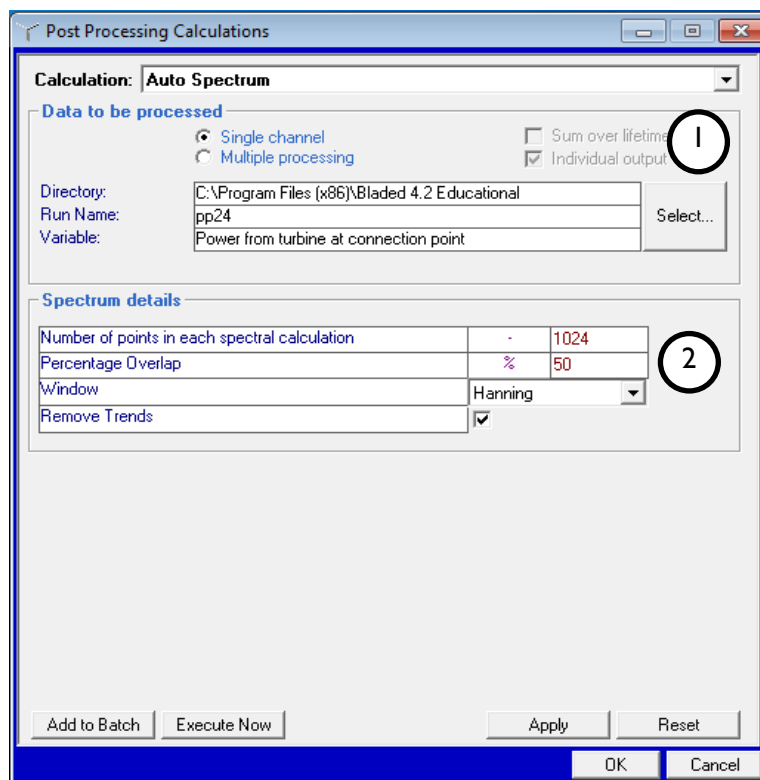


## Overview

Auto Spectrum performs a Fast Fourier Transform on a signal to calculate the Auto Spectral Density (frequency spectrum).

## Running the calculation

In the 'Post Processing' tab of the Calculations window, click on 'Auto Spectrum'. The following window will appear.



① By clicking 'Select' we can define the data for analysis by choosing the path, run name and data group. In the Educational version of Bladed we are only able to choose a single channel of data.

② Number of points in each spectral calculation must be entered as a power of 2. A larger value gives better frequency resolution, while lower values produce a smoother spectrum. 512 is a good starting point. The other inputs here can be entered as shown in the window above. (For a detailed explanation, press 'F1').

Click 'Execute now'. You will be asked where to save the results, and the calculation will run. Click 'Results' to open the Data View window. Define the data channel(s) to view results either graphically or in tabulated form in Excel.

# Channel Combination

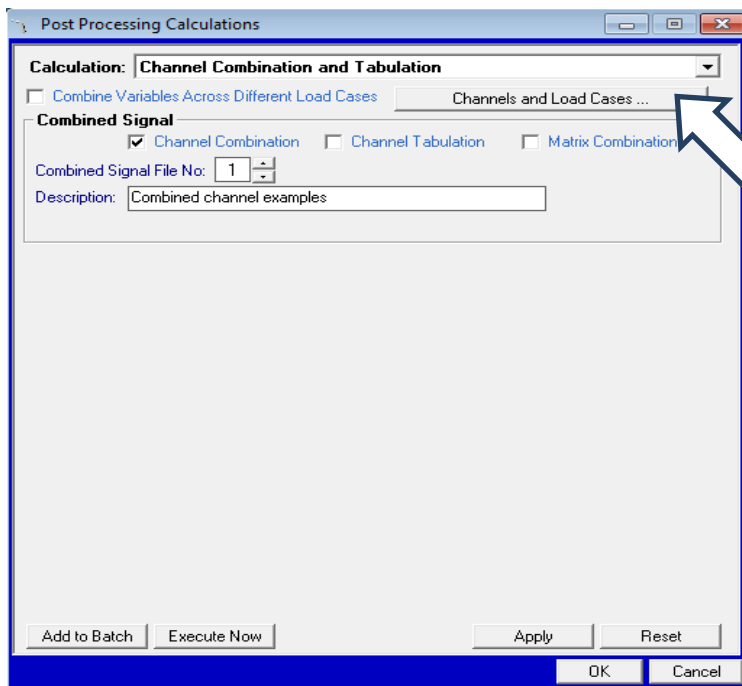
Channel Combination	
Flicker	
Linear Model	

## Overview

This allows new, user defined output channels to be created using combinations of existing channels and mathematical functions.

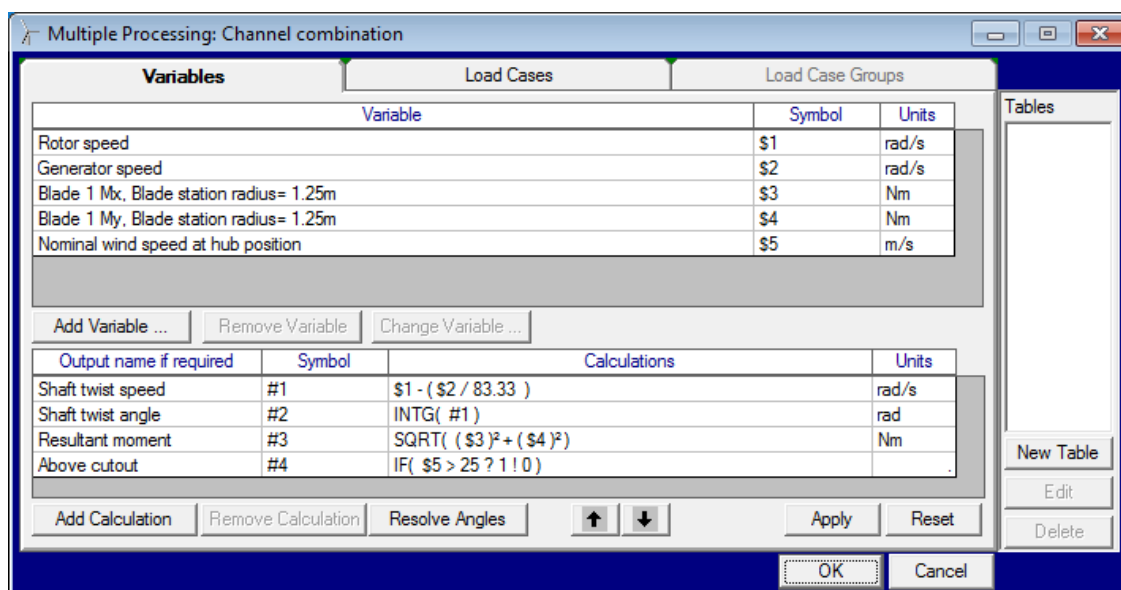
## Running the calculation

In the 'Post Processing' tab of the Calculations window, click on 'Channel Combination'.



The window shown to the left appears:

Click 'Channels and Load Cases' to open the 'Multiple Processing: Channel Combination' window shown below:

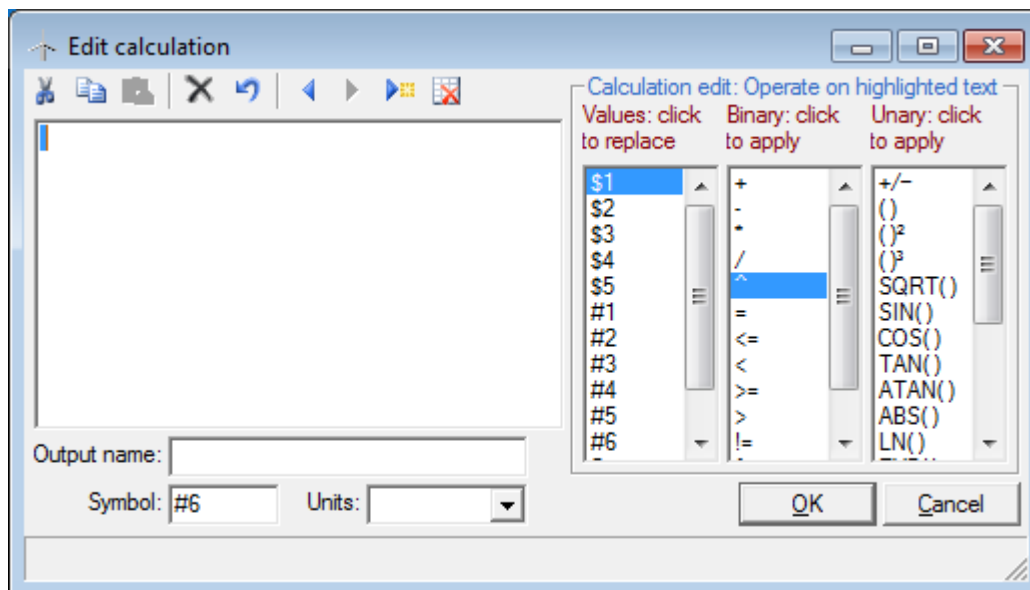


Choose the 'Load Cases' tab and add the directory and run name of the dataset to be processed.



Then choose the 'Variables' tab. Five variables are preselected for you. Click 'Add variable' to select any further variables for processing.

Equations for processing the variables can be set up by clicking 'Add Calculation'. The following window appears:



Create equations in the box on the left using the values and functions in the three menus on the right.

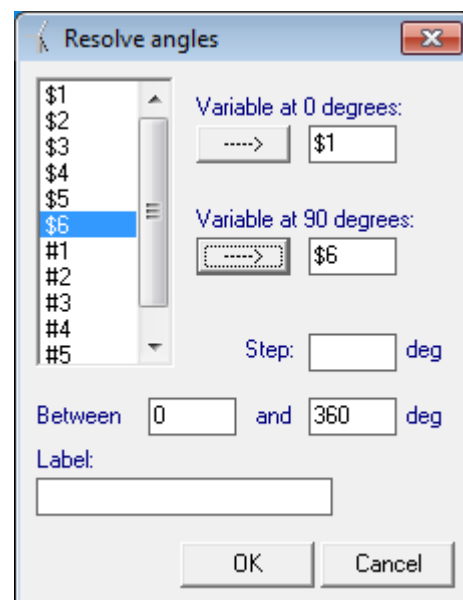
As well as defining new output channels, it is also possible to combine loads vectorially using the **Resolve Angle** function.

In the Multiple Processing: Channel Combination window, click on **Resolve Angles** to open the window on the right. This generates a set of equations which calculate the resultant of two orthogonal loads resolved into different angles.

Select the 0° and 90° loads from the list on the left of the window, using the arrow buttons to set them as variables. For example, the hub My and Mz loads could be selected.

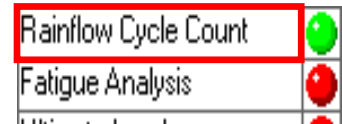
Then define a range of angles to resolve over, and provide a name for the output variable.

If the 0° and 90° loads are  $V_0$  and  $V_{90}$ , then for each angle  $\theta$  an equation is created which generates  $V_0 \cos\theta + V_{90} \sin\theta$ . The output name for each variable (if specified) will automatically have the appropriate angle appended.



For a more detailed explanation of this module press 'F1'.

# Rainflow Cycle Count



## Overview

The Rainflow Cycle Count post-processing calculation is a form of **fatigue loads** analysis, used to examine the results of time domain simulations. It can be used to find the **damage equivalent load**. This is done by counting the number of cycles of a signal, reducing the time variation of the load into a simple set of load means and load range values. This calculation is useful in analysing the total fatigue loading that a particular component is likely to experience during its lifetime.

## Running the calculation

1. In the 'Post Processing' tab of the Calculations window, click on 'Rainflow Cycle Count', and the following window appears.

The image shows a 'Post Processing Calculations' dialog box with the 'Rainflow Cycle Counting' calculation selected. The dialog is divided into three main sections: 'Data to be processed', 'Equivalent Loads Calculation', and 'Data Bins'. Section 1 (circled) is in the 'Data to be processed' section, showing 'Single channel' selected, 'Sum over lifetime' checked, and 'Individual output' unchecked. The directory is 'c:\program files (x86)\bladed 4.3 educational', the run name is 'pp24', and the variable is 'Rotating hub My'. Section 2 (circled) is in the 'Equivalent Loads Calculation' section, showing 'Calculate Equivalent Loads' checked, 'Inverse S-N slopes to use' set to 3, 4, 5, 6, 7, 8, 9, 10, and 'Frequency for Equivalent loads' set to 0.3 Hz. Section 3 (circled) is in the 'Data Bins' section, showing 'Automatic' selected, 'Number of bins' set to 60 (range 1-128), and 'Minimum range to count as a cycle' set to 0. At the bottom are buttons for 'Add to Batch', 'Execute Now', 'Apply', 'Reset', 'OK', and 'Cancel'.

① In the Educational version of Bladed we are only able to include a single channel of data for this calculation. By clicking 'Select' we can select the load case and variable we wish to analyse.

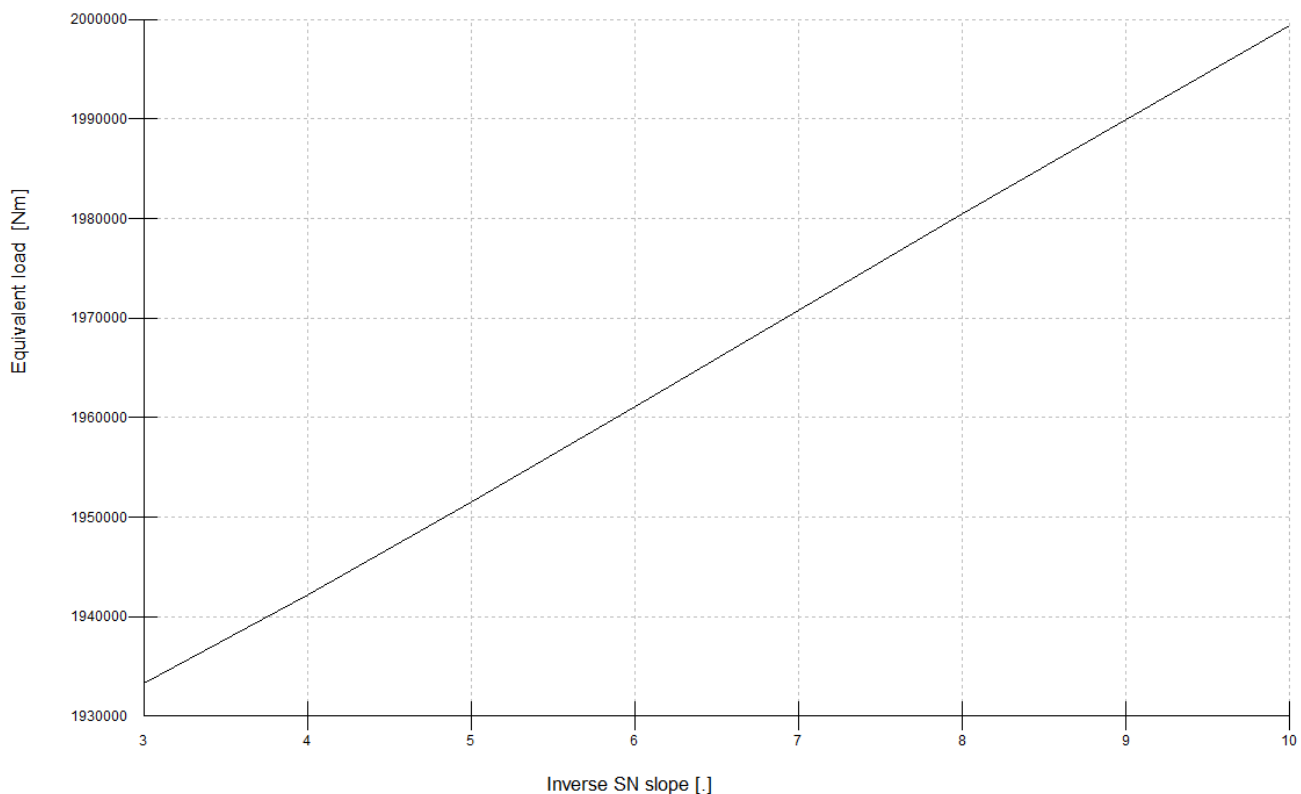
② Here we can choose to output an 'Equivalent Load'. Equivalent load values are derived from a hypothetical sinusoidal load that would produce the same fatigue load damage as the original simulated time series. The exact value of the equivalent load depends both on the sinusoidal frequency chosen and on the material specified. The S-N slope value is characteristic of a particular material and must be specified for each rainflow calculation. Up to 10 inverse S-N slope values can be entered.

③ We must also provide the number of bins and minimum range, as well as a minimum and maximum bin value if we so choose.

After the above information has been defined, we can run the calculation.

## Output and Applications

Once the simulation has been run, the values of the 'Equivalent Loads' can be viewed in the Data View panel. These values can be useful in the design process, and small adjustments can be made to the model to assess the effect this has on the value of these loads by re-running the calculation. Equivalent Loads of the same load case using different S-N slope values can also be analysed to assess the outcome of using different materials.



# Investigation Questions

The best way to become proficient with any new software is by using it.

The following study questions cover a range of key areas in turbine design. By using Bladed to answer them you will develop an excellent understanding of this sophisticated software package, and of wind turbine dynamics in general.

These questions could form the basis of an introductory course on Wind Turbine technology, and all of them can be investigated using the built-in demo turbine included with the software.

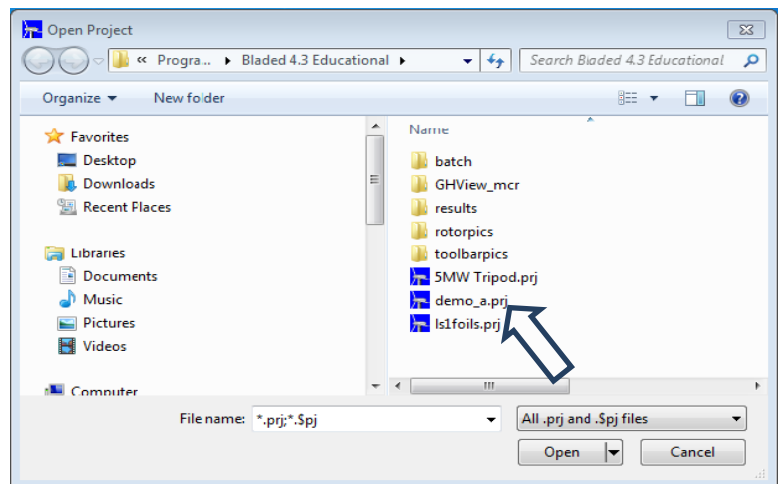
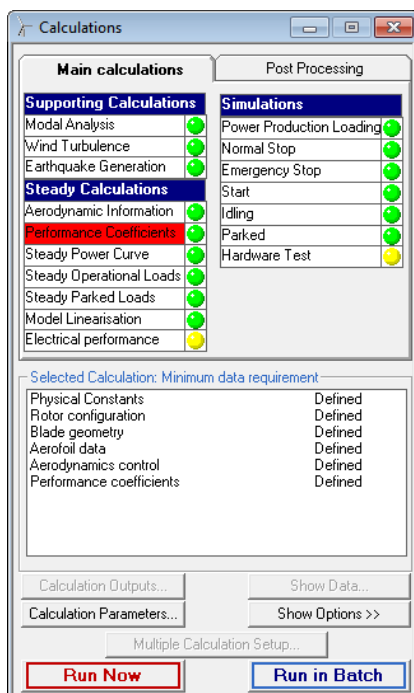
1. Which parameters affect the modal analysis of a turbine? Adjust blade mass and stiffness, tower mass and stiffness and nacelle mass and inertia. Run Modal Analysis and view the effect that these changes have on modal frequencies.
2. How does the number of blades affect turbine efficiency? Run steady power curve simulations with two and three blades. Note the differences in Power Output and Annual Energy Capture.
3. What effect does blade geometry have on turbine performance and loading? Experiment by changing the chord, thickness, and twist distribution. Run power production simulations and see how these changes affect turbine performance and structural loading.
4. What effect does blade flexibility have on output power and blade loads? Run simulations with and without blade modes enabled (use the 'Options' button on the calculation window to change these settings).
5. What effect does Tower Shadow have on the structural loading? Use Bladed to investigate the following loads: Blade Mx and My, Hub Mx and My, Tower My. Examine which of these loads are affected by Tower Shadow. Explain these results.
6. What effect does the wind shear profile have on the structural loading? Run power production simulations using steady wind with varying wind shear profiles. View the structural loading results. How does changing wind shear affect these loads?
7. What effect does the yaw angle have on power production and loading? Change the yaw angle by adjusting the wind direction. Run power production simulations and observe the differences in results.
8. Why is it preferable to vary the speed of the rotor at windspeeds below rated? Run power production simulations with different steady rotor speeds. Note the Annual Energy Yield each time, and also the performance coefficients.
9. Explain the difference between pitch regulated, stall regulated, fixed speed and variable speed control. Run Steady Operational Loads with each control option in turn. Examine the Electrical Power, Rotor Speed, Pitch Angle, Generator Torque and Blade and Hub loads as a function of wind speed. Explain the results.
10. How do the fatigue loads change when different S-N slope values or different frequencies for equivalent loads are applied? Run Rainflow Cycle Counting calculations on a turbulent wind power production simulation with different S-N slopes and frequencies. Note the variation in fatigue loads

# Worked example

## Q: How does changing the fine pitch angle affect Annual Energy Yield?

Most modern wind turbines use a variable speed / pitch regulated control strategy. Above rated windspeed the pitch angle of the blades is varied to maintain rated power. Below rated windspeed, the pitch angle is held constant and is called the fine pitch angle. To analyze the effect of fine pitch angle on the Annual Energy Yield (AEY), start by investigating AEY with the fine pitch angle as defined in the demo turbine (-2degrees). Then change the angle and recalculate AEY. In each case Optimal Mode Gain must be calculated. Detailed steps shown below.

1. Open the demo turbine.

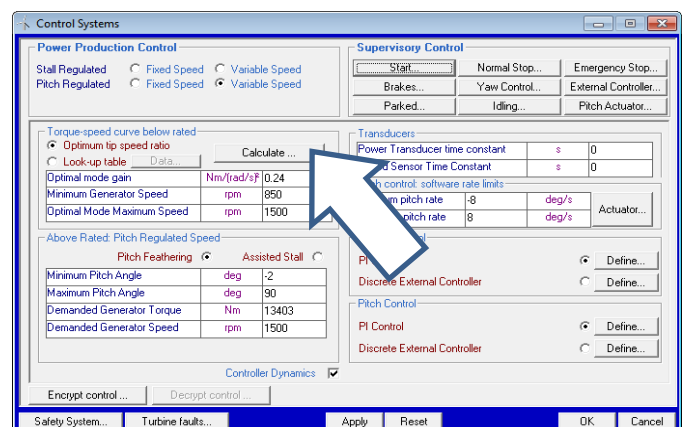


2. The first step is to calculate the Optimal Mode Gain. To do this, first run a 'Performance Coefficients' calculation. Open the calculation window and select 'Performance Coefficients' (as shown to the left). There is no need to change any of the calculation parameters at this stage.

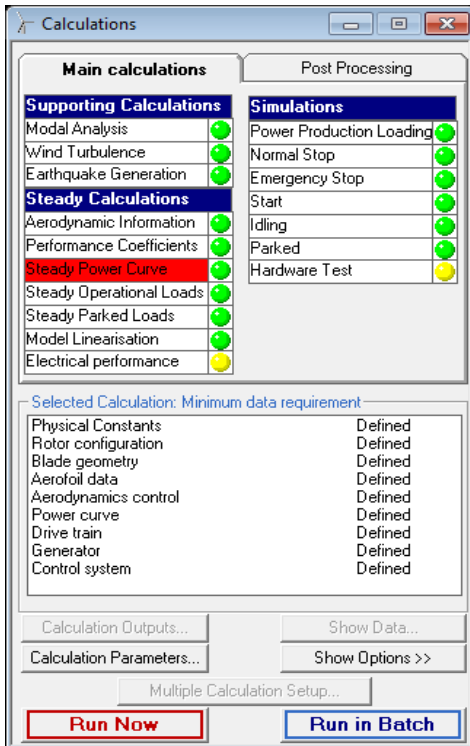
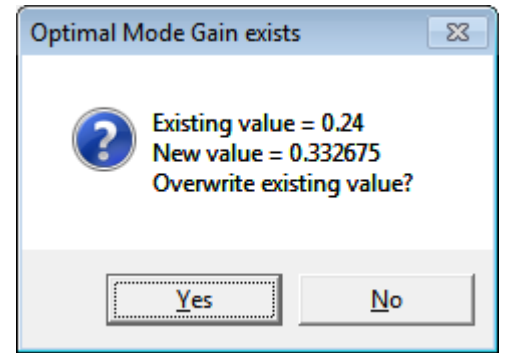
3. Click 'Run Now'.

4. Choose a file location to save the results. It is often best to create a new directory with a clear and specific name. This can be done by selecting the root directory and then writing the new file name at the end of the 'directory' path list (for example, you could create a file called \-2pitchangle). Leave the run name as it is ('pcoeffs' in this case). Click 'OK'. After the calculation is finished you can close the Calculation Progress window.

5. Next calculate the Optimal Mode Gain. Open the 'Control' window and click 'calculate'. You will be asked to confirm the air density and then to select your performance coefficients file.



6. You should see this window. Click 'Yes'. Then click 'OK' to close the Control Systems window.

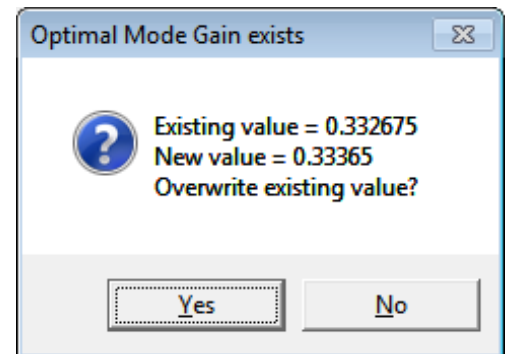


7. Now run a 'Steady Power Curve' calculation. Select 'Steady Power Curve' in the calculation window (as shown to the left), and click 'Run Now'. Save the result (with Run Name 'spcurve') in the same folder that you created in step 4.

8. Next, rerun these calculations with a new Fine Pitch Angle. To do this, first select 'Performance Coefficients' in the calculation window (as shown in step 2). This time, double click on 'Performance Coefficients' to open the Calculation Parameters window. Change the Pitch Angle from -2 to -1. Click 'OK'. Click 'Run Now' (this time, save the results in a new file '-1pitchangle').

9. Open the 'Control' module and recalculate optimal mode gain. You should see this window:

10. Click 'Yes'



11. You have now calculated a new Optimal Mode Gain for a fine pitch angle of -1. However, the control window still shows a 'Minimum Pitch Angle' of -2. So now change this value to -1. If you forget to do this, when you run a Power simulation, the controller will still define a minimum pitch angle of -2, while the Optimal Mode Gain is optimised for -1. This mismatch would mean reduced efficiency.

12. Now run a 'Steady Power Curve' calculation with your new pitch angle. This is the same as step 7, but save the 'spcurve' file in your '-1pitchangle' folder.

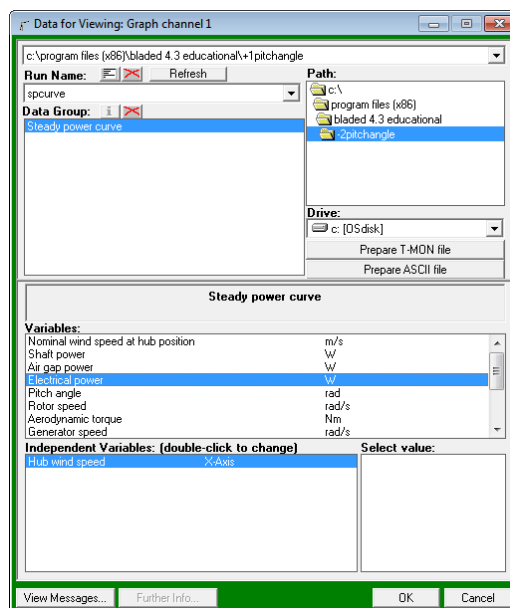
13. Repeat steps 8 to 12 for a fine pitch angle of 0, and also for +1. (Your new Optimal Mode Gain values should be 0.294304 and 0.248725 respectively.)

14. Open the 'DataView' module.

15. Click Channel 1 and select your '-2pitchangle\spcurve' file. Select Steady Power Curve, and choose 'Electrical Power' as the variable (as shown to the right).

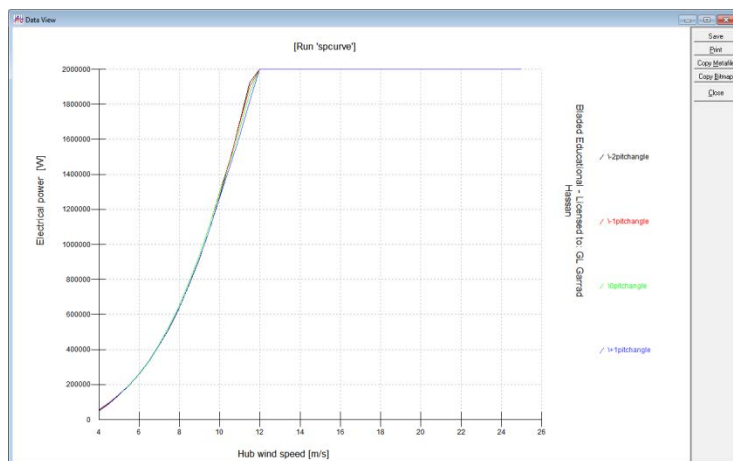
16. Click 'OK'.

17. Do the same in channels 2, 3 and 4, selecting 'Electrical Power' from your -1, 0 and +1 pitch angle files respectively.



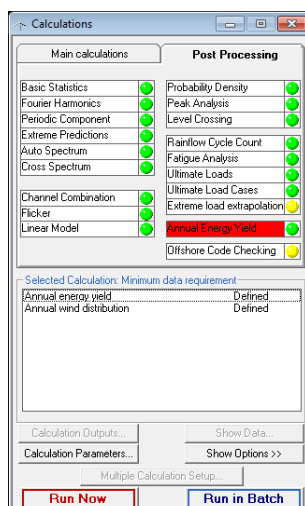
18. Click 'View Graph'.

Although the powercurves are very similar, there are small differences in power output at windspeeds below rated.

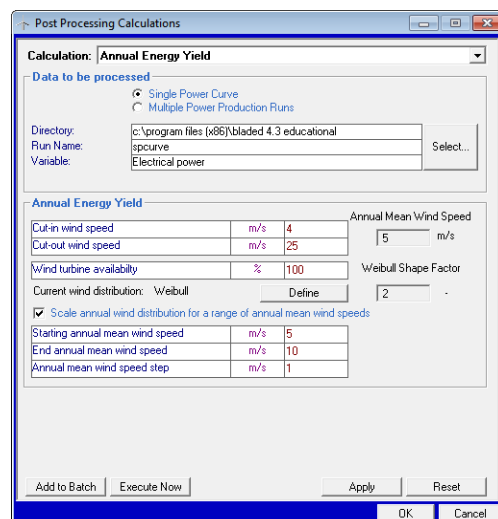


19. The final step is to compare Annual Energy Yield for the four different fine pitch angles.

20. Select 'Annual Energy Yield' in the Post Processing tab of the Calculation window.



Click on this to open the post processing window shown to the right:



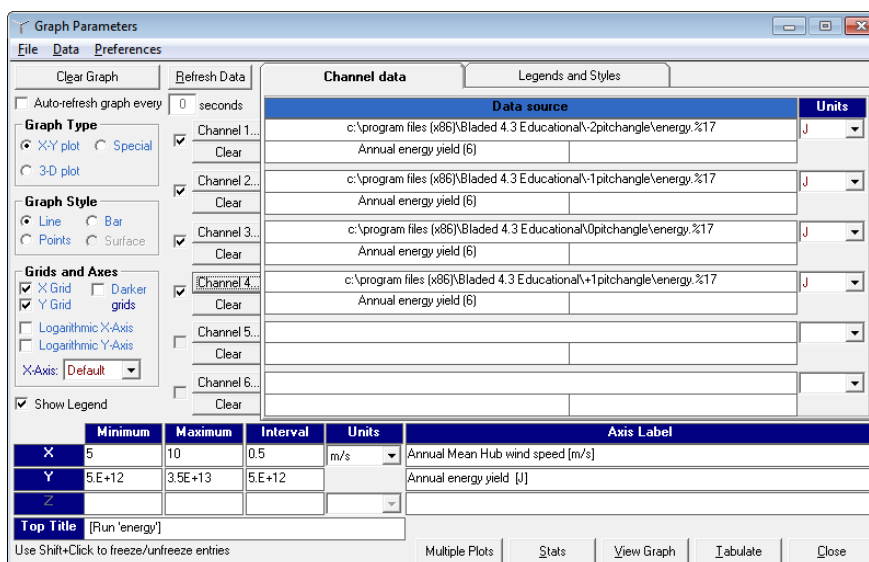
21. Click 'Select' to choose the data to be processed. Start by selecting your '-2pitchangle' spcurve file, with 'Electrical Power' as the variable. Click 'OK'.

For accurate Annual Energy Capture values you will need to define the wind distribution. However if you simply want to compare the relative Energy Capture of the different pitch angles, it is fine to leave the wind distribution with the default settings that already appear.

22. Click 'Execute Now' in the Post Processing Window. As always, select a sensible location to save your file. For example you could save the run in your \-2pitchangle folder, with a run name of 'energy'. Click 'OK', and run the calculation.

23. Repeat steps 21 and 22 for the -1, 0 and +1 pitch angle data.

24. Return to the DataView window. Use channels 1 to 4 to display the Annual Energy Yield for the four pitch angles (as shown below).



25. You could now click 'View Graph', but in this case it is probably better to display the data numerically by clicking 'Tabulate' at the bottom right of the Graph Parameters window. Choosing the 'Screen' option displays the data directly on the screen (as shown below).

The data shows that the optimum pitch angle from the four investigated is -1 degree.

Tabulated Data					
[Run 'energy']					
Annual energy yield [J]					
	Annual Mean Hub wind speed [m/s]	\-2pitchangle	\-1pitchangle	\0pitchangle	\+1pitchangle
1	5	8.55E+12	8.689E+12	8.682E+12	8.533E+12
2	6	1.404E+13	1.42E+13	1.417E+13	1.395E+13
3	7	1.98E+13	1.997E+13	1.991E+13	1.965E+13
4	8	2.524E+13	2.54E+13	2.533E+13	2.505E+13
5	9	3.001E+13	3.016E+13	3.008E+13	2.98E+13
6	10	3.393E+13	3.406E+13	3.398E+13	3.371E+13

**EXTENSION:** In step 5, Bladed calculates Optimal Mode Gain. Find out what formula Bladed uses to perform this calculation. Use the formula to manually calculate the Optimal Mode Gain values in this investigation. You could also have a go at deriving this formula from first principles.





