PHOENICS – Your Gateway to CFD Success

Documentation for PHOENICS |TR 316

FLAIR-EFS User Guide.

CHAM Ref: CHAM/TR316
Document rev: 6
Doc. release date: 09 October 2022
Software version: PHOENICS 2022 v1.0

Responsible author: J C Ludwig
Other contributors: M Camps Santasmasas, D R Glynn
Editor: J C Ludwig
Published by: CHAM

Confidentiality:
Classification: Unclassified

CHAM, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK
Telephone: 020 8947 7651 Fax: 020 8879 3497
E-mail: phoenics@cham.co.uk Web site: https://www.cham.co.uk

CHAM – Your Gateway to CFD Success

FLAIR-EFS User Guide: TR 316

Table of Contents

1. Foreword

PHOENICS/FLAIR is a CFD software package specialising in the simulation of scenarios involving fluid flow, heat transfer, combustion and chemical reaction processes occurring in the built and natural environment. FLAIR is utilised by architects, design engineers and safety officers concerned with the performance of air-flow systems for both the internal and external environment.

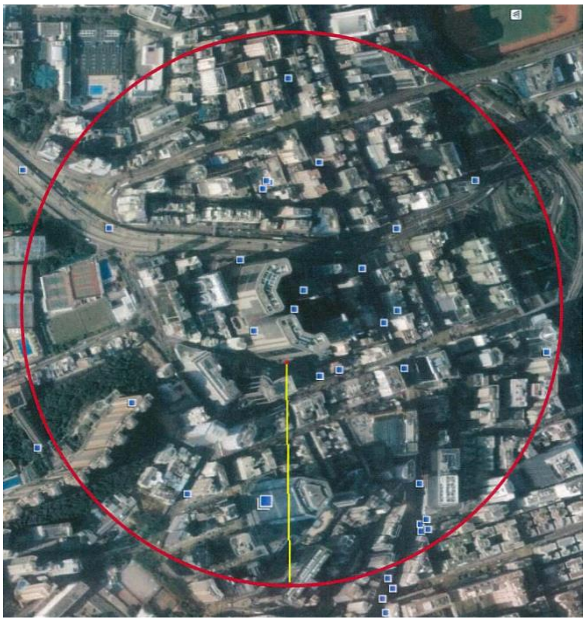
Based upon the VR Menu used by PHOENICS/FLAIR, **FLAIR-EFS** is a reduced-cost subset for users concerned only with the external environmental conditions surrounding buildings and other structures.

The function of **FLAIR-EFS** is to simulate the air flow, temperature distribution and pollutant concentration around individual or groups of buildings.

It will predict:

- Forces on the exterior of buildings, roofs and walls.
- Pedestrian comfort information including:
    - Wind Amplification Factors.
    - Probability of the wind speed to exceed a set threshold value
    - NEN8100 Dutch standard for Wind Comfort and Wind Danger in the Built Environment.
    - Other pedestrian wind comfort standards can be implemented.
- Rates of heat loss or gain between buildings, atmosphere and sky.
- Dispersion and concentration of pollutants.

FLAIR-EFS intended for architects, building engineers, urban planners, local authorities and environment engineers. **FLAIR-EFS** enables users to visualise, understand, evaluate and refine the air-flow patterns in steady-state or time-dependent scenarios, in micro- as well as macro-scale.



FLAIR-EFS offers:

- CAD import and repair features.
- Grid generation with refinement in the region of interest.
- Wind and wind profiling.
- Solar gain.
- Interface to weather databases (e.g. Energy Plus)

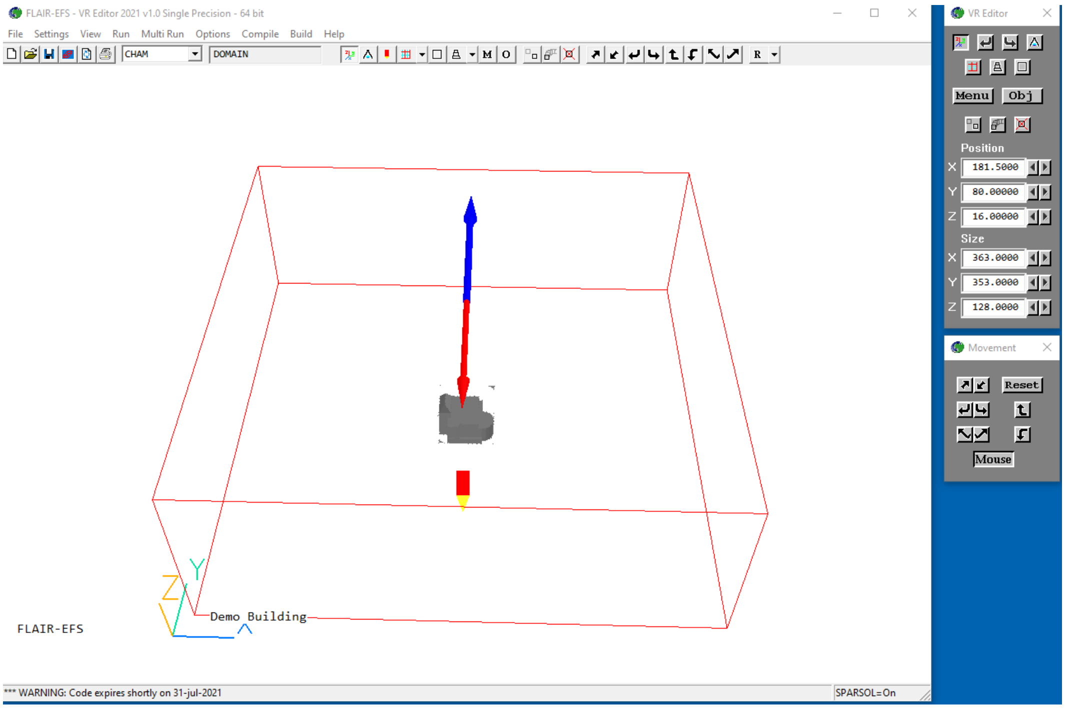
2. Running FLAIR-EFS

2.1 Getting Started

FLAIR-EFS can be started by:

- Clicking the FLAIR-EFS icon (a desktop shortcut created by the FLAIR-EFS installation program); or
- Start the VR-Editor by clicking on 'Start', 'Programs', 'FLAIR-EFS', then 'FLAIR-VR'.

In either case, the VR screen shown in the figure below should appear



At the top of the main window is the **Top Menu bar**, starting 'File', 'Settings' etc. Immediately below that is the **Tool Bar**, containing standard icons for file manipulation. These are described on-line here. Under the main graphics window is the **Status bar**, showing the current working directory. To the right of the main window are the **handset** and **motion control** panels, which are described on-line here.

2.2 Getting Help

When the cursor rests for more than 3 seconds on a panel button in the handset or an icon on the **toolbar**, then a description of the function of that button or icon appears in a small text box ('bubble-help') next to the item.

The 'Help' item on the top menu bar (last on right) contains links to this document, a basic tutorial and further documentation on PHOENICS.

2.3 Starting a New Case

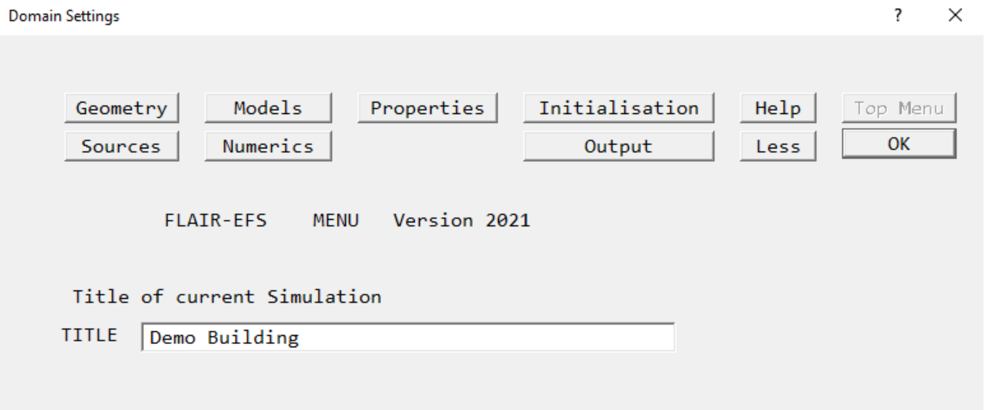
To start a new piece of work, you need to start from the default FLAIR-EFS setup, then modify this as required. On the top menu bar click 'File' then 'Start New case' or click the 'Start new case' icon on the toolbar. From the list of applications offered (this will depend on the licence file) select 'FLAIR-EFS' and click OK. The current setup will be deleted and the default case loaded. The screen should look as above.

Notice that the default geometry is a single building. The wind (shown by the red arrow) is blowing from the North (shown by the blue arrow). The case contains the following objects:

- WIND – (of type WIND), it sets the wind speed and direction.
- BUILDING – (of type BLOCKAGE). It represents the building around which the wind flows.
- BUILDZONE – (of type NULL). It represents the region of interest, initially the same size as the building.

3. FLAIR-EFS Main Menu

The Main menu is accessed by clicking the M icon on the tool bar, or the Menu button on the handset. The FLAIR-EFS Main Menu dialog is shown below. The precise menu date may change!



The buttons along the top of the panel allow you to set and modify your case. In general, it is best to start at the top left, and work from left to right, as this minimises the chances of missing out settings.

The buttons perform the following functions:

- Geometry: Geometry and grid settings.
- Models: Solution of variables, turbulence models, comfort indices etc.
- Properties: Density, viscosity etc.
- Initialisation: Initial values.
- Help: Help on the current panel.
- Top menu: Go to the top level.
- Sources: Whole-domain sources, e.g. buoyancy, Coriolis.
- Numerics: Solution control settings.
- Output: Print-out and field dumping controls.

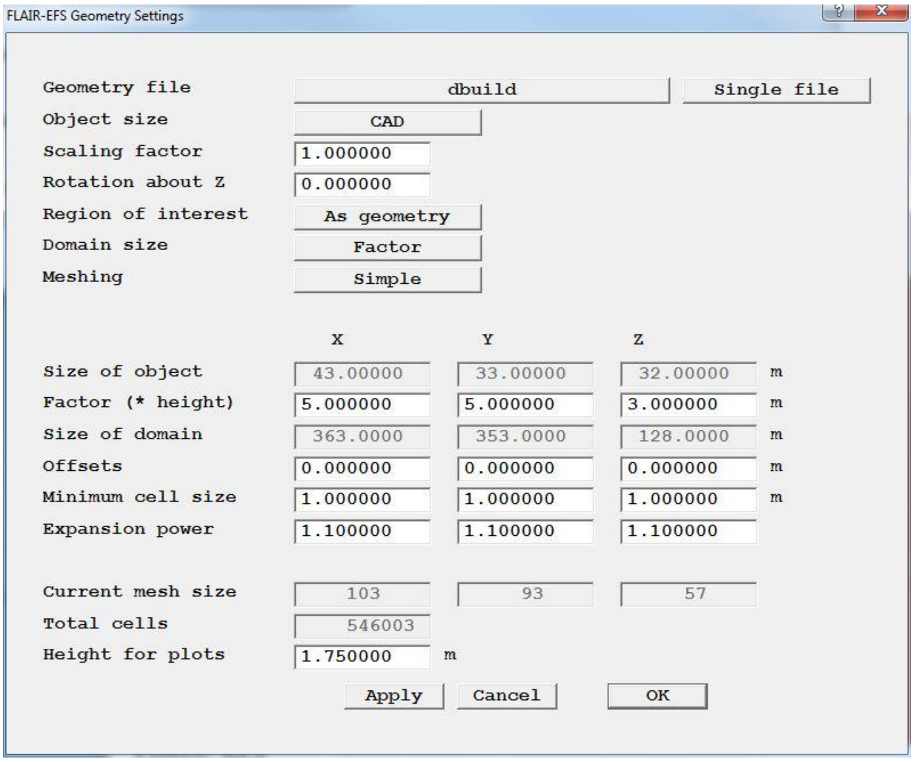
The (show) 'Less' button is special, in that when pressed it changes to (show) 'More', and all the menu panels show only the most commonly used options. This mode is aimed at beginners and in-frequent users, who do not wish to be swamped with options they may not need. All the images of menu panels are in the full (show) 'more' mode. The mode is recorded in the Q1 file saved at the end of the session, and hence will be used next time the Q1 is loaded.

Each of these panels will now be described in turn.

4. Geometry Creation

4.1 Geometry

The FLAIR-EFS Geometry Settings dialog controls the shape(s) of the building(s) considered, and also the computational mesh used to solve the equations. The default dialog, reached by clicking '**Geometry**' on the **Main Menu**, is shown below:



4.1.1 CAD File Formats

The first step is to create a building complex or a city scape in your accustomed way and save it using one of the following CAD formats:

- STL - Stereo lithography file available in many popular CAD programs as an export format.
- 3DS - Autodesk 3ds Max
- WRL - Virtual Reality Modelling Language file
- DW - Files generated by DesignWorkshop from Artifice
- AC - Files generated by AC3D from Inivis
- DXF - Drawing Exchange Format File (AutoCAD)
- IV - Files generated by Open Inventor

4.1.2 Importing a Single CAD File

To import a CAD file containing a single building, or indeed multiple buildings defined in a single CAD file, click the button next to **Geometry file** (labelled **dbuild** in the image above). A file browser will appear. Navigate to where the CAD file is, select it, and click 'Open'. The selected CAD file will be imported, and the dialog will update to show the name of the new file. The '**Size of object**' and '**Size of domain**' fields will also update, based on the size of the new object. The '**Size of object**' defines the size of the geometry region referred to in Section 4.3.

Note that the file browser will only allow a single file to be selected. Each import operation will replace the current file with the new one.

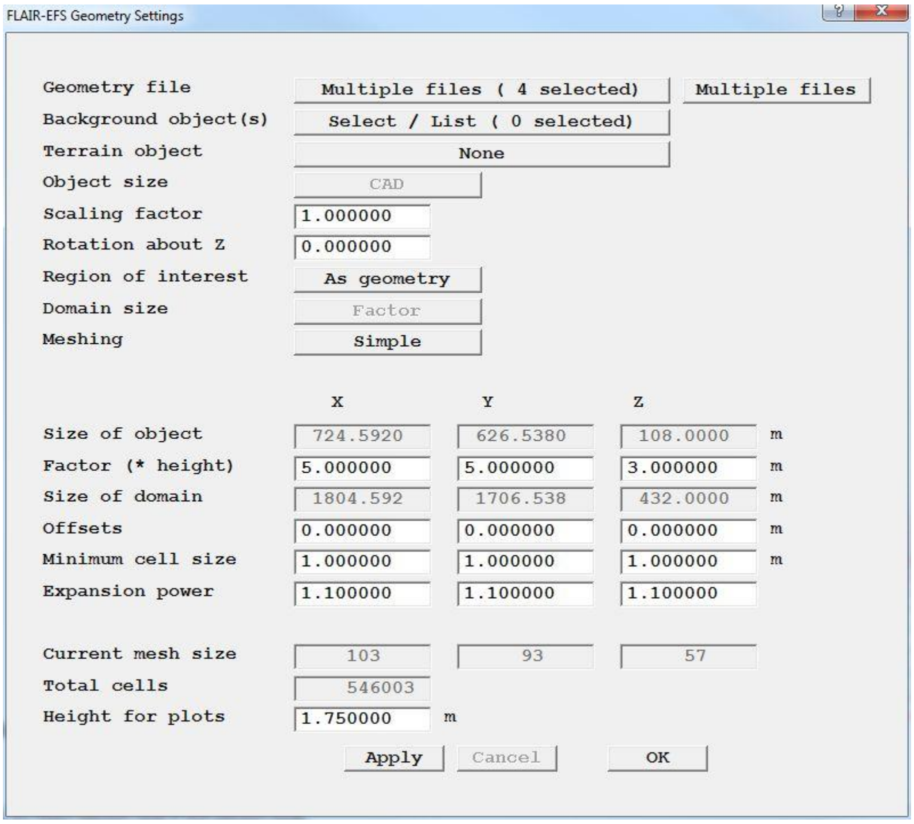
4.1.3 Importing Multiple CAD files

If the geometry to be modelled consists of several buildings held in several CAD files, click the button labelled '**Single file**'. It will change to '**Multiple files**'. Proceed as above, clicking the button next to '**Geometry file**'. The file browser which opens will now allow multiple files to be selected at the same time using standard Windows selection methods. The files do not need to be of the same type.

When you have selected all the files, click 'Open'. The selected files will be imported. The dialog will update to show 'Multiple files selected'. The '**Size of object**' field will update to show the size of the smallest bounding box which encloses all the imported shapes, and this is used to update the '**Size of domain**' field.

Note that the coordinates held in the CAD files must use a common origin, otherwise the parts will not appear in the correct relative positions.

Once multiple files have been imported, the Geometry Settings dialog will change to something like this:

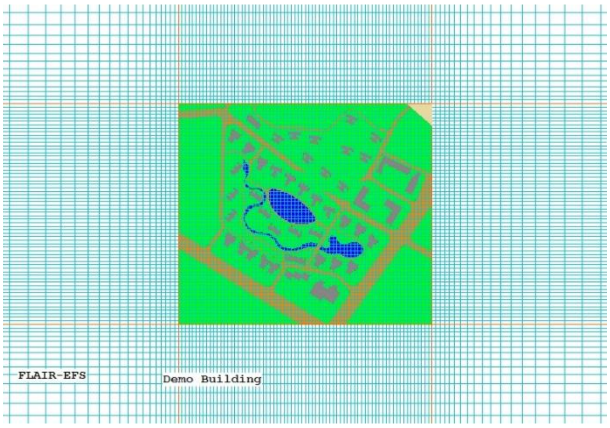


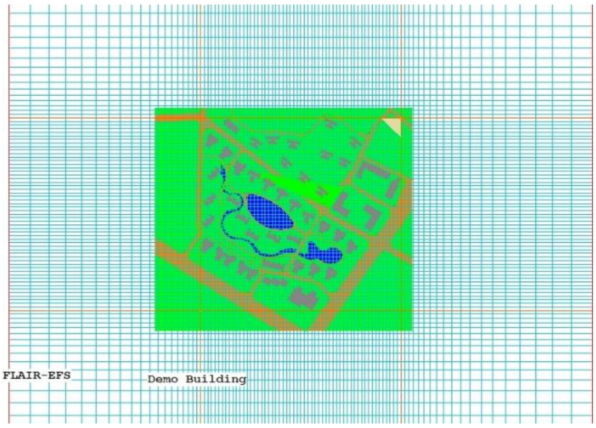
4.1.4 Background Objects

By default, all the objects imported through the '**Multiple files**' option described above will be considered when evaluating the size of the minimum bounding box, shown on the dialog as '**Size of object**'.

In the following images, on the left none of the objects have been designated 'background, so the geometry region fits the biggest object – the green one.

On the right, the green, blue and orange objects are designated 'Background', so the geometry region fits the grey object – the buildings. As explained in Section 4.3, a further region of interest can be created within the geometry region.





4.1.5 Terrain Object

One of the multiple objects in the scene may represent the terrain. By selecting it, the height used in the Wind Amplification Factor (WAF) calculations (See Section 10.1) will be taken as the height above the upper surface of this object rather than above Z=0.0.

If this object extends to, or beyond, the edges of the domain, it should be marked as a 'background' object.

4.1.6 Scaling and Rotating the Geometry

FLAIR-EFS operates in the SI unit system, so all dimensions are in meters. If the imported CAD file uses a different unit system (e.g. mm, cm, ft, in), then the geometry must be scaled accordingly. Enter the scaling factor in the input box next to '**Scaling factor**' and click '**Apply**'. The scaling operation is cumulative, so scaling by 0.1 and then by 10 will restore the original size.

The size of the imported object(s) is by default taken from the CAD file. To manually set the size, click the button labelled '**CAD**' next to '**Object size**'. The button label will change to '**User-set**', and the entry-boxes for the object size will become active.

The size of the object(s) is taken to be the size of the smallest bounding box which will exactly enclose all the geometry.

By default it is assumed that the Y axis points North, and that the Z axis points up. The alignment between Y and North can be changed on the Wind Attributes dialog described later (Section 5.5). If necessary, the imported geometry can also be rotated about Z so that it correctly aligns with North.

4.2 Size of the Domain

The size of the domain can be set as factors of the object height (**Domain size – Factor**), or be a user input (**Domain size – Total**). When set to '**Factor**', the domain size is set as:

Xsize_domain = Xsize_object+2*Xfactor*Zsize_object

Ysize_domain = Ysize_object+2*Yfactor*Zsize_object

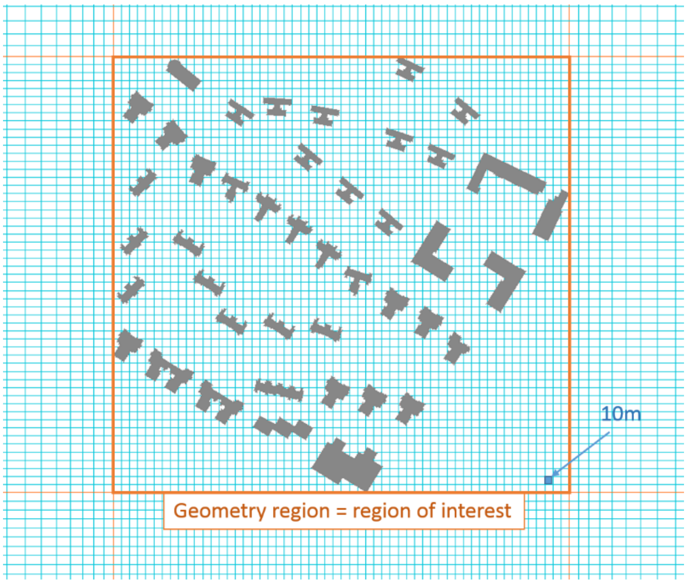
Zsize_domain = Zsize_object+1*Zfactor*Zsize_object

These sizes are echoed in the greyed-out '**Size of domain**' input boxes, and are updated when '**Apply**' is clicked. The building zone is then centred on the domain.

The X, Y or Z factors can be set to zero, in which case the domain is the same size as the building zone.

4.3 Regions of Interest

The main region of interest is the building zone, the entire zone covering the imported geometry as shown below:

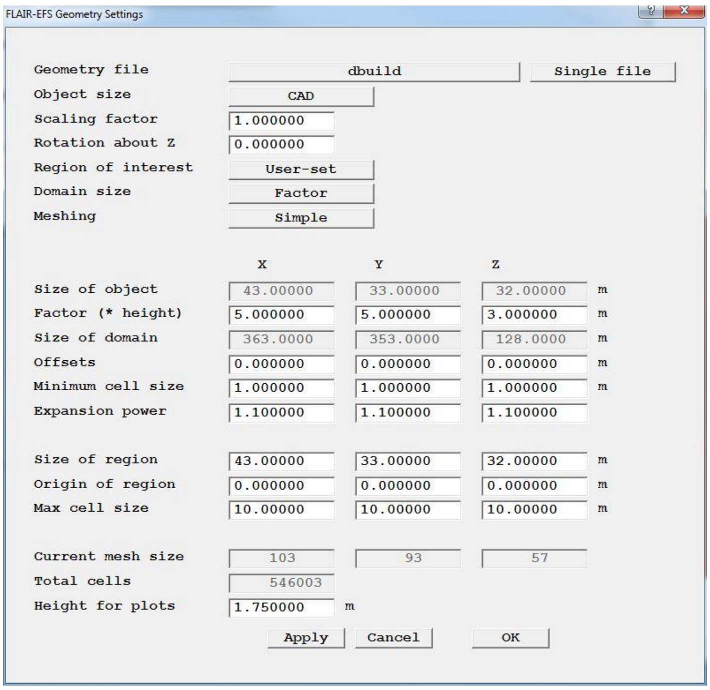


Note that in the above image, the region of interest exactly coincides with the outer bounding box of the buildings. To allow extra space between the edges of the geometry and this bounding box, the '**Offsets**' can be set greater than zero. The domain size will not change.

Offsets	0.000000	0.000000	0.000000	m
Minimum cell size	10.00000	10.000000	1.000000	m

The 10m mesh size marked in blue above is the result of setting the '**Minimum cell size**' to 10 in X and Y.

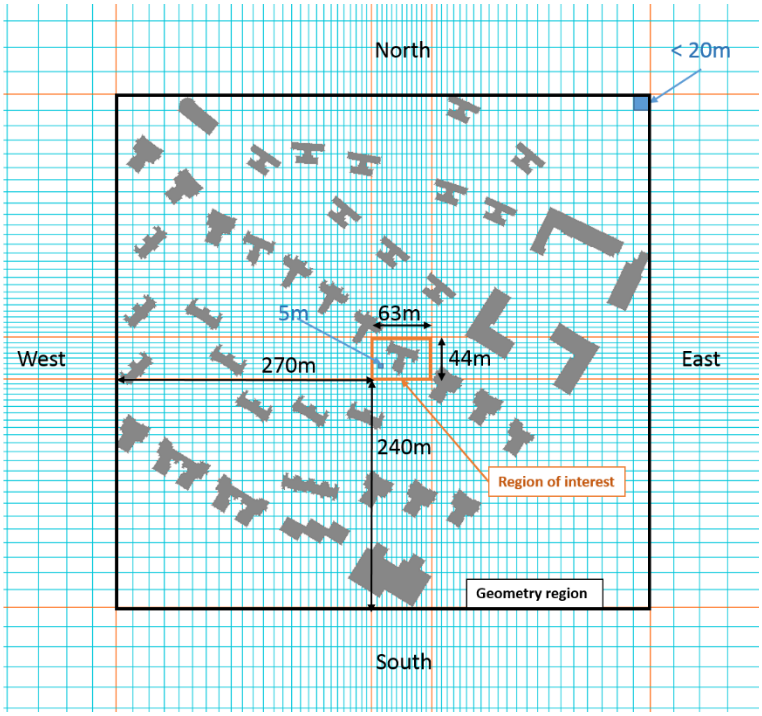
A second, inner region of interest can be created by toggling '**Region of interest**' from '**As geometry**' to '**User-set**'.



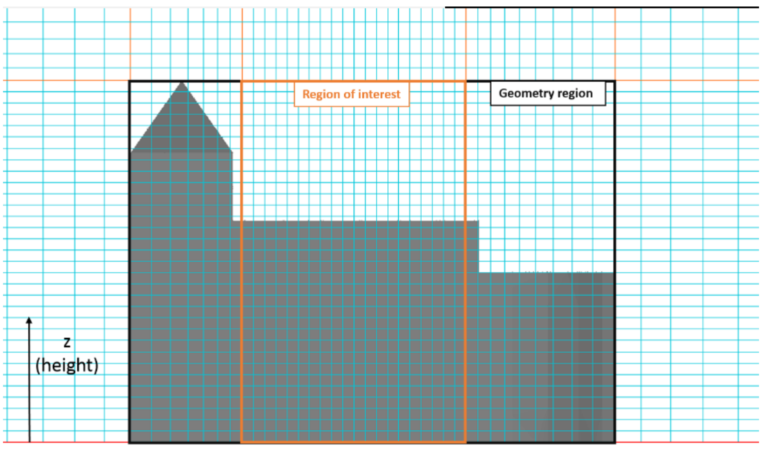
For example, the following settings:

Minimum cell size	5.000000	5.000000	1.000000	m
Expansion power	1.100000	1.100000	1.100000	
Size of region	63.00000	44.00000	108.0000	m
Origin of region	270.0000	240.0000	0.000000	m
Max cell size	20.00000	20.00000	10.00000	m

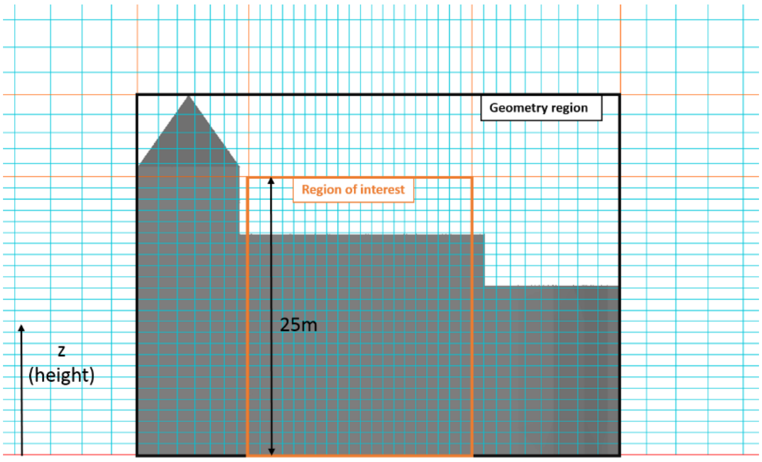
Result in the zones shown below. Note that the ‘**Origin of the region**’ is relative to the origin of the buildings zone, not the domain.



If the Z-direction size of the region of interest is set to 0, the full height of the building zone will be used, as shown here:



Otherwise, the value entered is used to set the Z size:



The Z origin of the region of interest can be greater than zero if required.

4.4 Meshing

As has already been mentioned, and as is visible from the above images, the solution mesh is controlled from the Geometry Settings dialog.

- If '**Meshing – Simple**' is selected, the number of cells is controlled from this dialog.
- If '**Meshing – PHOENICS**' is selected, the meshing is controlled by the normal PHOENICS mesh generator. This allows more detailed control over the number of cells and spacing in each direction, but requires more input (and knowledge) from the user. Once PHOENICS meshing is selected, the dialog changes to the PHOENICS Grid Mesh Settings dialog, with the same grid as in FLAIR-EFS. The Grid Mesh Settings dialog is described in the main PHOENICS documentation TR326 here. To switch back to FLAIR-EFS meshing, click the EFS Meshing – PHOENICS button. Any changes made in the PHOENICS mesh dialog are lost when switching back, and the EFS grid will be re-instated.

For '**Simple**' meshing, the number of cells is controlled by:

- The '**Minimum cell size**'. If there is no separate region of interest ('Region of interest – As geometry'), the number of cells in the geometry region is the size of the geometry region divided by the minimum cell size.
- If there is a separate region of interest ('Region of interest – User-set'), the number of cells in the region of interest is the size of the region of interest divided by the minimum cell size. Outside the region of interest, the grid expands geometrically towards the edge of the geometry region, initially with the set '**Expansion power**', with the constraint that the expansion power may be reduced until the biggest cell is smaller than the '**Max cell size**'.
- Outside the geometry region the grid expands geometrically towards the domain boundary using the set '**Expansion power**'.

The current mesh size in each coordinate direction, and the total number of cells are displayed at the bottom of the dialog.

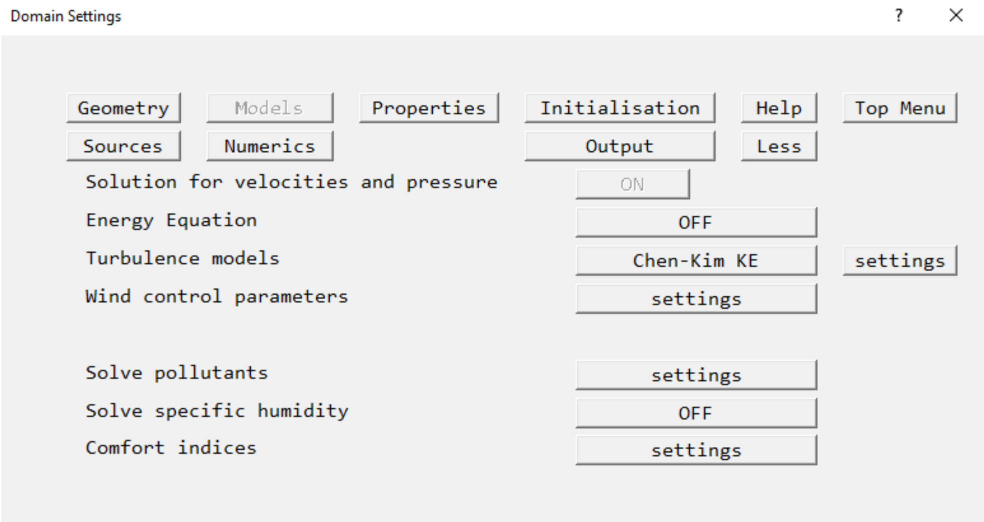
To decrease the number of cells:

- Make the 'Minimum cell size' bigger. This will make the solution in the inner zone less accurate.
- Make the 'Max cell size' bigger. This will make the solution in the outer building zone less accurate
- Make the 'Expansion power' bigger. This will make the solution in the outer-most zone less accurate.

To increase the number of cells, or improve accuracy, make the above settings smaller. The more cells used, the greater the chance of accurate results at the cost of increased runtime.

5. Models

The Models panel looks like this:

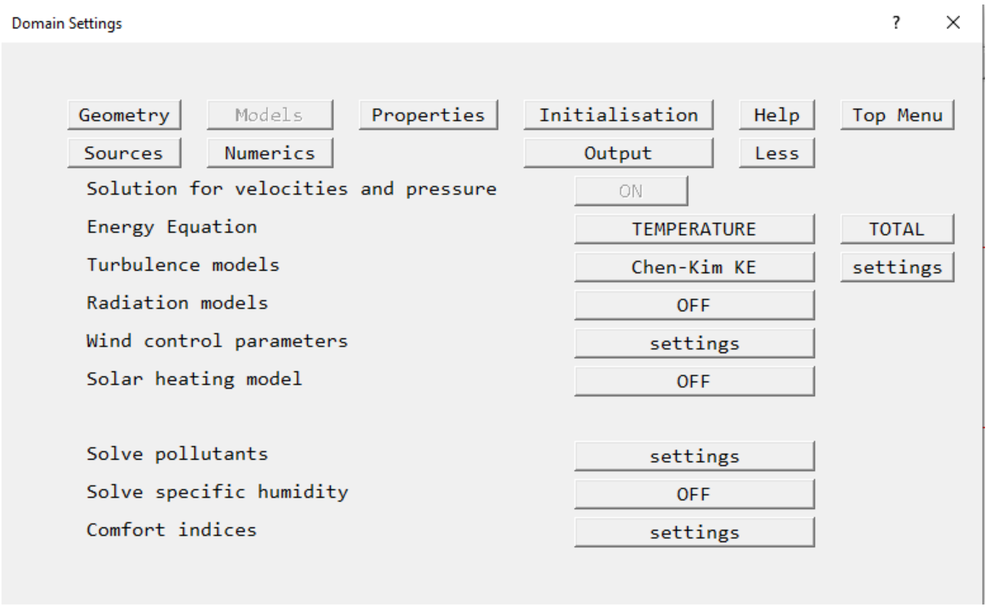


5.1 Solution for velocities and pressure

FLAIR-EFS always solves for velocity and pressure. This cannot be turned off.

5.2 Energy Equation

By default FLAIR-EFS solutions are iso-thermal. If temperature effects are important, then solution of the energy equation should be turned on. Once this happens, further options become available:



5.3 Turbulence models

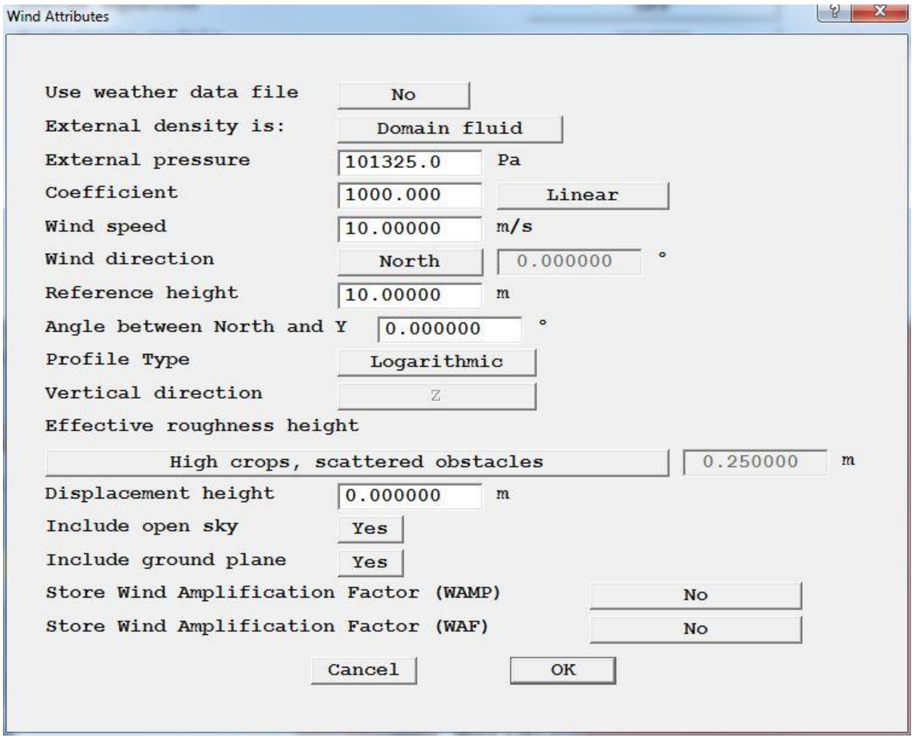
Here the turbulence model used is chosen. The default KECHEN model will be suitable for most cases.

5.4 Radiation models

This option is only available if the Energy Equation is ON. The IMMERSOL or P1T3 models can be turned on. The models are described in the PHOENICS Encyclopaedia here and here respectively.

5.5 Wind control parameters

This displays the attributes dialog of the WIND object. It can also be reached by double-clicking the WIND object on the screen.



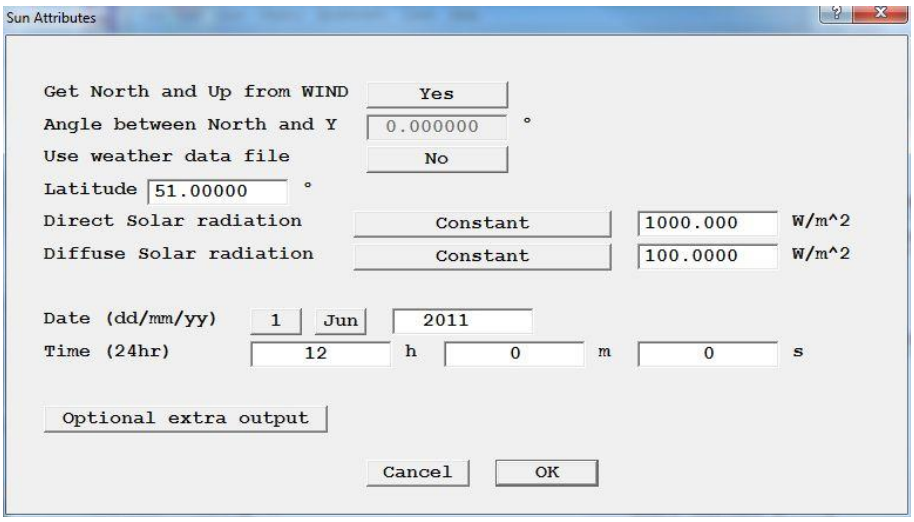
From here you can set:

- The wind speed, direction and reference height.
- The surface roughness.
- Optionally access a weather data file.

A full description of the dialog is given on-line here.

5.6 Solar heating model

This option is only available if the Energy Equation is ON. When turned ON, a SUN object is created. The 'Settings' button which appears leads to the attributes dialog of the SUN object, shown here:



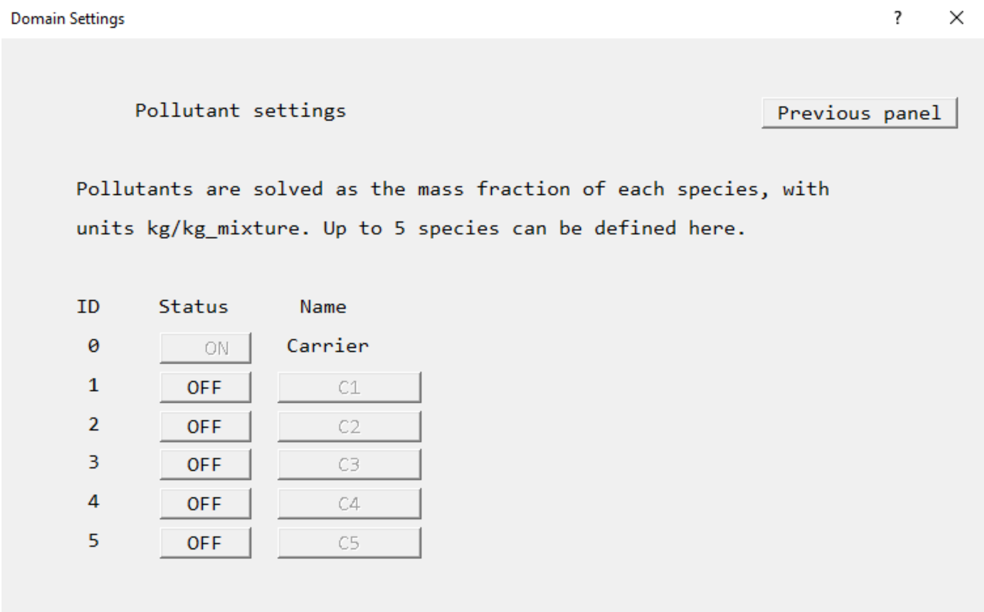
From here you can set:

- The latitude where the building is located.
- The date and time of day.
- The direct and indirect solar heating.
- Optionally access a weather data file.

A full description of the dialog is given on-line here.

5.7 Solve pollutants

The ‘Settings’ button leads to this dialog:

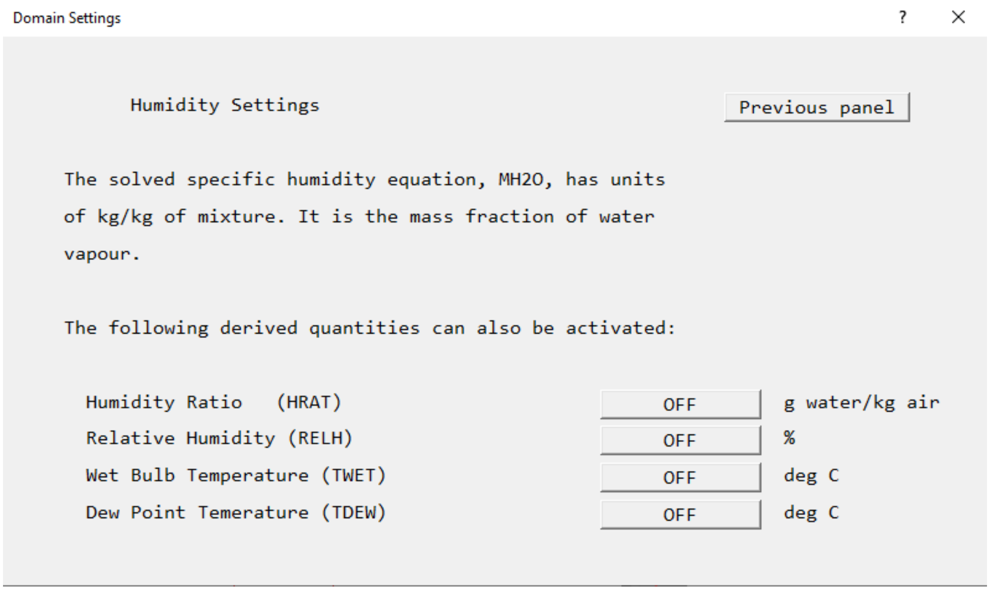


From here you can activate the solution of up to five passive pollutant species. If the Energy Equation is ON, you can choose to include the pollutant concentration in the gas density calculation. In that case, you will need to supply the molecular weight of each species. A source (or sources) of pollutants will also have to be specified.

5.8 Solve Specific Humidity

If the 'Solve Specific humidity' button is switched to 'On', the specific humidity equation, MH2O, will be solved. The variable MH2O has units of kg water vapour/kg mixture. It is a mass fraction of water vapour.

The ‘Settings’ button leads to this dialog:



5.8.1 Boundary Condition settings

The units used to specify boundary sources (inlets, openings, volume sources at BLOCKAGE objects, area sources at PLATE objects) can be set on the dialogs for the individual objects.

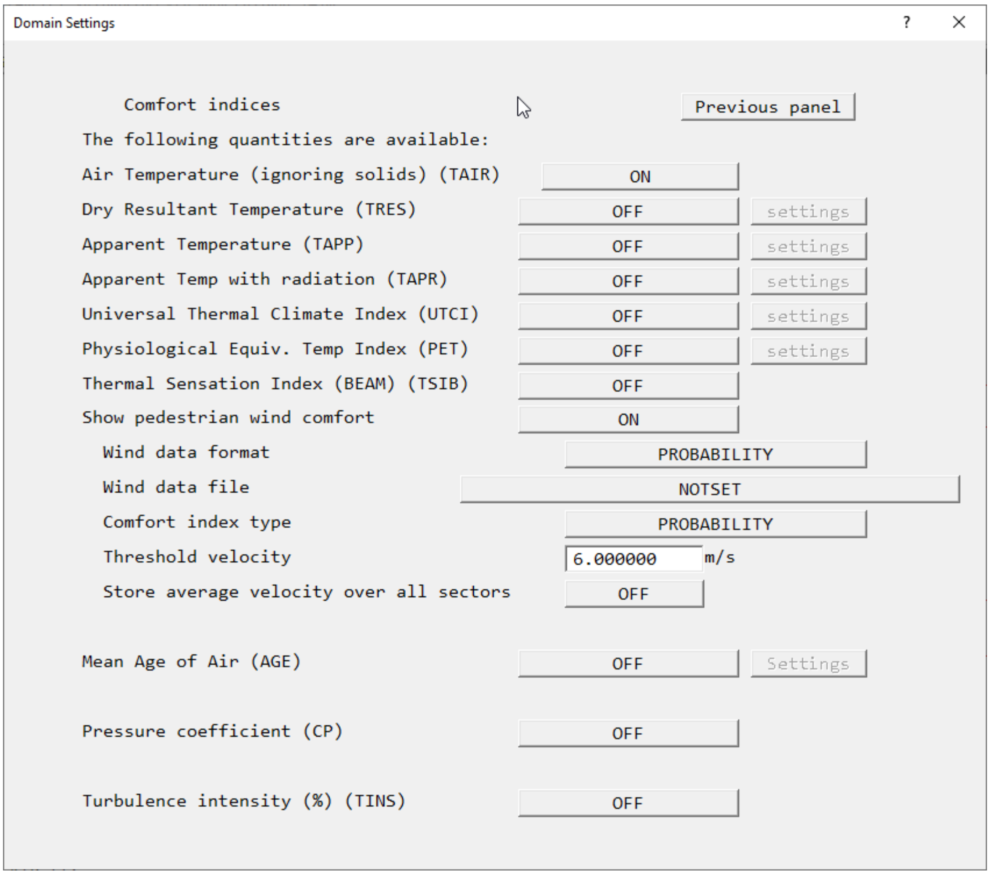
The options are:

- Specific Humidity (Mass fraction) (kg/kg, kg/s, kg/s/m^3, kg/s/m^2)
- Humidity ratio (g/kg, g/s g/s/m^3, g/s/m^2)
- Relative humidity (%) (at inlets and openings)

The default is Specific Humidity.

5.9 Comfort indices

The ‘Settings’ button leads to this dialog:



5.9.1 Dry Resultant Temperature

The dry resultant temperature is a standard index used to show the level of comfort within the occupied space. It is a function of air temperature, air velocity and mean radiant temperature. The formula as defined in Volume A of the CIBSE Guide is:

$$Tres = (Trad + Tair * (10*vel)^{0.5}) /(1+ (10* vel)^{0.5})$$

Where
Tres = the resultant temperature;
Trad = mean radiant temperature;
Tair = air temperature (TEM1 in PHOENICS-Flair);
vel = air velocity (taken to be the local VABS in PHOENICS-Flair).

The mean radiant temperature can be a user-set constant value, or can be taken to be the T3 radiation temperature of the IMMERSOL radiation model.

Comfortable values of Tres are typically in the range 16-28 deg C, depending on the external conditions and type of occupancy.

5.9.2 Apparent Temperature

Apparent Temperature is a general term for the perceived outdoor temperature, caused by the combined effects of air temperature, relative humidity, radiation and wind speed.

The formulae for the Apparent Temperature used in PHOENICS-Flair are those used by the Australian Bureau of Meteorology. They are an approximation of the value provided by a mathematical model of heat balance in the human body. They can include the effects of temperature, humidity, wind-speed and radiation.

Two forms are given by the Australian Bureau, one including radiation and one without.

The expression without radiation is suitable for the working condition of people walking in the shade; the expression with radiation is suitable for the working condition of people in direct sunlight.

The TAPP Apparent Temperature is the non-radiation version. The expression used is:

$AT = Ta + 0.33 \times e - 0.70 \times ws - 4.00$

where:

Ta = Dry bulb temperature (C) (TEM1 in PHOENICS-Flair)
e = Water vapour pressure (hPa) [humidity]
ws = Wind speed (m/s) at an elevation of 10 meters (taken to be the local VABS in PHOENICS-Flair)

The vapour pressure e is calculated from the temperature and relative humidity using the equation:

$e = rh / 100 \times 6.105 \times \exp (17.27 \times Ta / (237.7 + Ta))$

where:

rh = Relative Humidity [%] (RELH in PHOENICS).

The variable name used to store the non-radiative AT is TAPP.

The TAPR Apparent Temperature includes the radiation term. The expression used is:

$AT = Ta + 0.348 \times e - 0.70 \times ws + 0.70 \times Q/(ws+10.0) - 4.25$

where:

Q = Net radiation absorbed per unit area of body surface (w/m2)

and the other terms are as defined above.

Steadman, 1973[1] indicates that Q is likely to be in the range -20 to 130 W/m2.

The variable name used to store the AT is TAPP.

Note that in order to calculate either Apparent Temperature, the solution of Specific Humidity and the derivation of Relative Humidity must both be turned on.

5.9.3 Universal Thermal Climate Index

The Universal Thermal Climate Index UTCI provides an assessment of the outdoor thermal environment in bio-meteorological applications based on the equivalence of the dynamic physiological response predicted by a model of human thermoregulation, which is coupled with a state-of-the-art clothing model. The operational procedure, which is available as software from the UTCI website (www.utci.org), shows plausible responses to the influence of humidity and heat radiation in the heat, as well as to wind speed in the cold and is in good agreement with the assessment of ergonomics standards concerned with the thermal environment.

The necessary research for this was conducted within the framework of a special commission of the International Society of Biometeorology (ISB) and European COST Action 730.

The published subroutine returns the UTCI as a function of air temperature, water vapour pressure, mean radiant temperature and wind speed 10m above ground level. In the Flair implementation, the local air temperature is taken to be the solved temperature TEM1, the local water vapour pressure is derived from the solved water vapour mass fraction MH2O or from a user-set constant, the local mean radiant temperature is taken as the solved radiant temperature T3 or a user-set constant, and the wind speed is taken to be the local absolute velocity VABS.

5.9.4 Physiological Equivalent Temperature

The PET comfort index, derived from the human heat balance model, combines weather and thermo-physiological parameters (clothing and human activities). It is used to measure the thermal comfort of an individual in a given situation by comparing their physiological responses to those they would have in the reference environment, for example an office in which they feel generally comfortable.

For the study of indoor spaces, the PET index is often used as its reference environment is comparable to an office and it is easier to apply and less theoretical than the Universal Thermal Climate Index (UTCI).

The published subroutine (https://palm.muk.uni-hannover.de/trac/browser/palm/trunk/SOURCE/biometeorology_pet_mod.f90?rev=3467&order=author) returns the PET index as a function of air temperature and pressure, water vapour pressure, mean radiant temperature and wind speed 1.1m above ground level. In the Flair implementation, the local air temperature is taken to be the solved temperature TEM1, the local water vapour pressure is derived from the solved water vapour mass fraction MH2O or from a user-set constant, the local mean radiant temperature is taken as the solved radiant temperature T3 or a user-set constant, and the wind speed is taken to be the local absolute velocity VABS.

Other assumed inputs are for a standing male aged 35, height 1.75m, weight 80kg, metabolic rate 80W (1.4mets, sedentary, or light standing activity) and clothing insulation 0.9clo.

5.9.5 Thermal Sensation Index

The Thermal Sensation Index (TSI) is an empirical model developed by studies conducted in the context of Japan. It considers five climatic factors— air temperature, horizontal solar radiation, wind speed, relative humidity and mean radiant temperature— at a given location and time. TSI's prevalence in Hong Kong can be explained by its user-friendly nature. Particularly, it is the index that public housing projects in Hong Kong adopt.

TSI is subdivided into a scale of seven levels, where level 1 is cold, and level 7 is hot. While level 4 is considered neutral, a TSI between 3 and 5 is deemed thermally acceptable, where no major discomfort is expected.

TSI can be calculated by the following formula:

TSI = 1.7 + 0.1118Ta + 0.0019SR – 0.322WS – 0.0073RH + 0.0054ST

Where,

Ta = air temperature (°C)
SR = horizontal solar radiation (W/m2)
WS = wind speed (m/s)
RH = relatively humidity (%)
ST = surrounding ground surface temperature (°C), assumed to be Ta+3°C for TSI calculation

We have implemented a simplified version according to BEAM Plus New Buildings Version 2.0[2]. In the simplified BEAM implementation, the following values are assumed constant:

Ta = air temperature (°C) = 31°C
SR = horizontal solar radiation (W/m2) = 254W/m2 in shadow, 525W/m2 in the sun
WS = wind speed (m/s) = VABS as calculated by PHOENICS-Flair-EFS

RH = relatively humidity (%) = 72%
ST = surrounding ground surface temperature (°C), assumed to be Ta+3°C for TSI calculation

5.9.6 Show Pedestrian Wind Comfort

When set to ON, as shown above, modelling of pedestrian wind comfort is enabled.

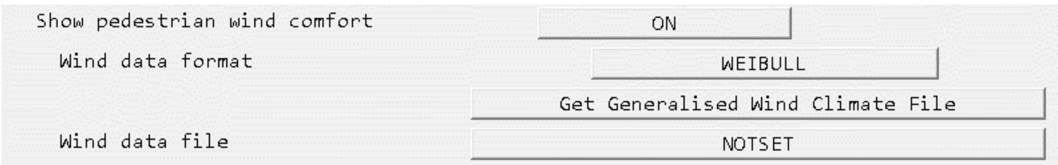
CFD studies of wind flow around buildings may be required to produce pedestrian wind comfort parameters, based on statistical meteorological data. This will typically require running a number of cases, with wind directions from various points of the compass, and then using the statistical data to generate probability-weighted average velocities.

The statistical data is input into FLAIR or FLAIR-EFS in the form of a “Wind Data file”, which is typically based on wind data obtained over a period of several years. This specifies the probability of the wind having a particular speed and a particular direction. More specifically - the total range of possible wind speeds is divided into a specified number of intervals, and the 360° range of possible wind directions is divided into a specified number of sectors. Then for every speed interval and every sector, a probability value is provided.

To activate the wind averaging, set **'Store average velocity over all sectors'** to ON. The **'Wind data file'** button allows you to browse to the appropriate file.

Wind data format'. The wind statistics can be represented in one of two ways:

- Frequency tables in WAsP (.tab) histogram format
- Weibull coefficient tables. These can be taken
    1. directly from a Generalised Wind Climate(GWC) file (WAsP libfile), or
    2. a file containing just the coefficients for the required height and roughness.



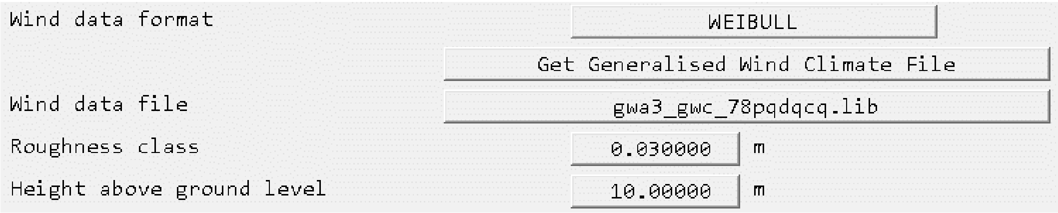
The **'Get Generalised Wind Climate File'** button opens the Global Wind Atlas website. From here, wind data for large parts of the world can be downloaded in the form of Generalised Wind Climate Files.

Once the required area has been selected, download the GWC file from the 'Customised Areas' tab on the right of the screen. The file will appear in the Downloads area, with a .lib extension.

From the **'Wind data file'** button browse to the downloaded file and select it. It may be convenient to copy or move the GWC file to the current working directory before opening it.

If the data format is selected as WEIBULL, and the data file specified has the file extension '.lib', then it will be treated as a GWC file. If the extension is anything else, it will be assumed that the file contains the exact coefficients already.

If a GWC file is detected, additional buttons appear:



to allow the specification of the Roughness class (Possible Values are: 0.00, 0.03, 0.10, 0.40, 1.50 m), and Height above ground (Possible values are: 10, 50, 100, 150 and 200 m) available in all GWC files.

An example and detailed specification for the WAsP and Weibull coefficient inputs is given in the Appendix.

Note that in the Attributes panel of the Wind Object you should set the wind direction as usual, but you do NOT set the wind speed; this is defined automatically to be the weighted average for the given direction over the range of wind speeds, based on the probabilities in the data table. Likewise, when **'Store average velocity over all sectors'** is ON, the reference height for the wind profile is automatically set to the measurement height specified in the Wind data file, and so does not need to be set by the user.

Each of the runs should be saved "as a case". After the last run has been made, the output files of all the runs must be passed through a utility program called PHISUM, which does the necessary averaging and produces statistical data based on the given wind frequencies. PHISUM is activated from "Run" / "Utilities" on the top bar of the VR Editor, and will ask the user to specify the file names of the ".pda" files containing the results of the simulations for each sector modelled. The values of average velocity over the wind speeds and sectors are stored in the variable VAV in the output file from PHISUM.

Contour plots of the average wind velocity VAV may be generated using the Viewer. When entering the Viewer the "User-set file names" option should be selected. For "Input file", any one of the individual-sector Q1 files should be specified. For "Solution file", the output PHIDA file from PHISUM should be specified.

In addition to the average velocity, three statistical parameters can be plotted from the output file of the PHISUM program, to generate various assessments of pedestrian comfort. These are selected by **'Comfort index type'**.

- **'Probability of exceeding'**: This is the probability, between 0 and 1, for the wind speed to be higher than the user-set **'Threshold velocity'**. The variable name for plotting in VR-Viewer is PRO.
- **'NEN8100'**: Dutch standard for Wind Comfort and Wind Danger In the Built Environment shown in the table below. Please note that the standard uses letters to name each probability bin instead of the numbers and colours used by **FLAIR-EFS**. The variable name for plotting in VR-Viewer is NEN.

Wind comfort					
P (U_{THR} > 5m/s) (in % hours per year)	Grade	Colour	Activity		
			Traversing	Strolling	Sitting
< 2.5	1		Good	Good	Good
2.5 - 5.0	2		Good	Good	Moderate
5.0 - 10.0	3		Good	Moderate	Poor
10.0 - 20.0	4		Moderate	Poor	Poor
> 20.0	5		Poor	Poor	Poor

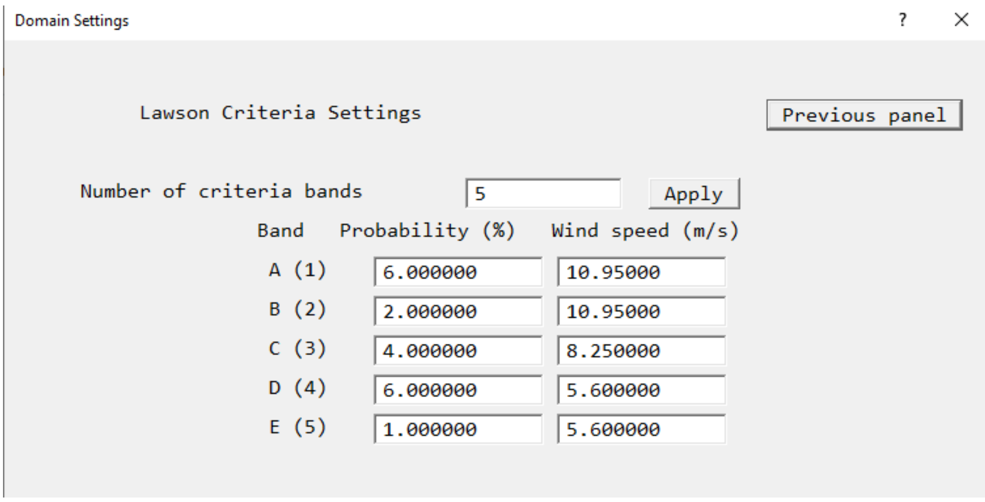
Wind danger			
P (U_{THR} > 15m/s) (in % hours per year)	Grade	Colour	Risk
0.05 -0.3	6		Limited risk
> 0.3	7		Dangerous

- **'Lawson Criteria'**: Lawson is similar to NEN, but uses a different classification. The Lawson Comfort Criteria specify a range of pedestrian activities, and for each activity define a wind speed and maximum frequency of exceedance. If the wind speed exceeds the threshold for the activity, then the conditions are unacceptable. The default criteria are:

Activity	Band	Probability	Threshold Wind Speed
Roads and Car Parks	A -> 1	6%	10.95 m/s
Business walking	B ->2	2%	10.95 m/s
Pedestrian walk-through	C -> 3	4%	8.25 m/s
Pedestrian standing	D -> 4	6%	5.6 m/s
Sitting	E -> 5	1%	5.6 m/s

The categories are ordered, from the least comfortable to the most comfortable. The band indicates the highest-number activity for which the conditions will be acceptable. Thus for example, if at a certain location condition (5) was met, then conditions would be deemed comfortable for sitting, hence band 5. But in a more consistently windy location only condition (1) might be met, hence band 1, meaning that so far as pedestrian comfort is concerned this area is suitable only for roads or car parks. Plots showing the Lawson bands are frequently found in CFD assessment reports for proposed building developments. The variable name for plotting in VR-Viewer is LAWS. The variable names for the individual probabilities of exceedance for the individual bands are PRO1, PRO2,...,PRO$_{Ncrit}$ where Ncrit is the number of criteria bands.

The above values of threshold wind speed and exceedance probability for the Lawson criteria appear to be in fairly widespread use. However, different values are seen in some publications. Therefore, an option to change the values has been provided; this is accessed by clicking the Settings button for Lawson.



Up to 6 threshold velocity / probability bands can be specified.

5.9.7 Mean Age of Air

This quantity represents the time since entry at each point in the domain. The units are seconds.

In 'dead' zones, such as in recirculation areas, the time since entry will tend to a large value as the air will be trapped there. These values should be treated as indicative rather than exact. In regions where there is a reasonable exchange of air, the values will be correct. The variable name for plotting in VR-Viewer is AGE.

5.9.8 Pressure coefficient

This is the local pressure divided by the dynamic head at the wind reference velocity. The variable name for plotting in the VR-Viewer is CP.

5.9.9 Turbulence intensity

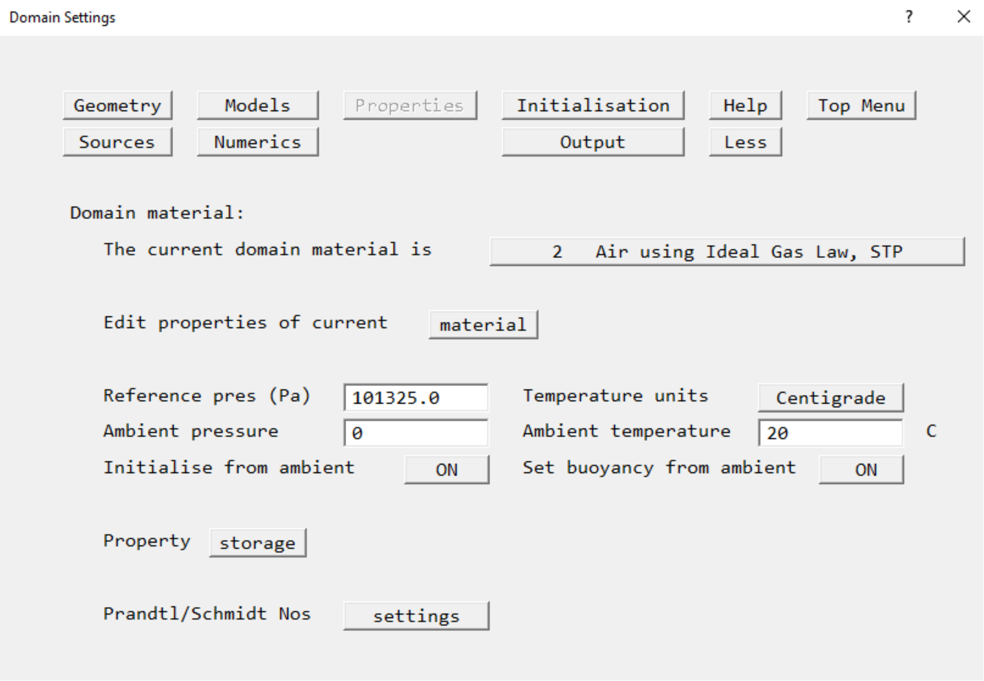
This is a measure of the strength of the local turbulent fluctuations. It is defined as:

$I = 100*k^{1/2}/V_{abs}$

Where k is the turbulent kinetic energy and V_{abs} is the local wind speed. The variable name for plotting in the VR-Viewer is TINS.

6. Properties

The properties panel (with Energy Equation ON) is shown below.

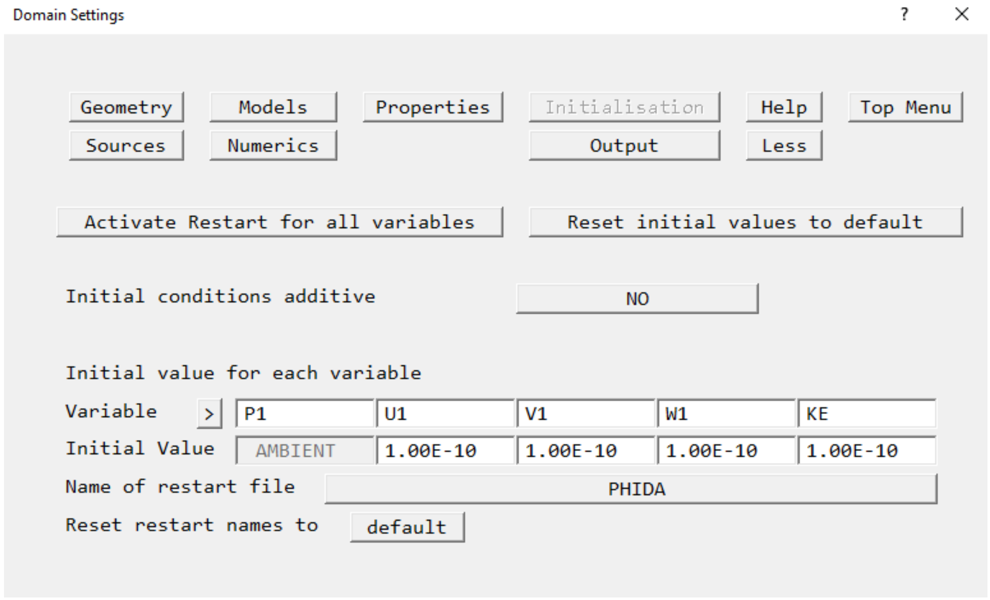


From this panel you can:

- Set the fluid properties. By default, the fluid flow through the domain is air. If the Energy Equation (Section 5.2) is OFF, the properties of air at 1 atmosphere and 20˚C are used. If it is ON, the density is calculated from the Ideal Gas Law, thus allowing for density variation due to pressure and temperature changes. If the solution of pollutants (Section 5.7) is active, a mixture molecular weight can be used.
- Set the reference pressure and temperature units (Centigrade or Kelvin). If the domain were located at high altitude, for example, the reference pressure should be reduced to the local air pressure.
- Set the ambient pressure and temperature. The ambient pressure is relative to the reference pressure. These values are used as the initial condition for the solution, and as the default external conditions. If the domain is located in a region of high or low temperature, the ambient temperature should be set to the prevailing local temperature.

7. Initialisation

The initialisation panel is shown below.

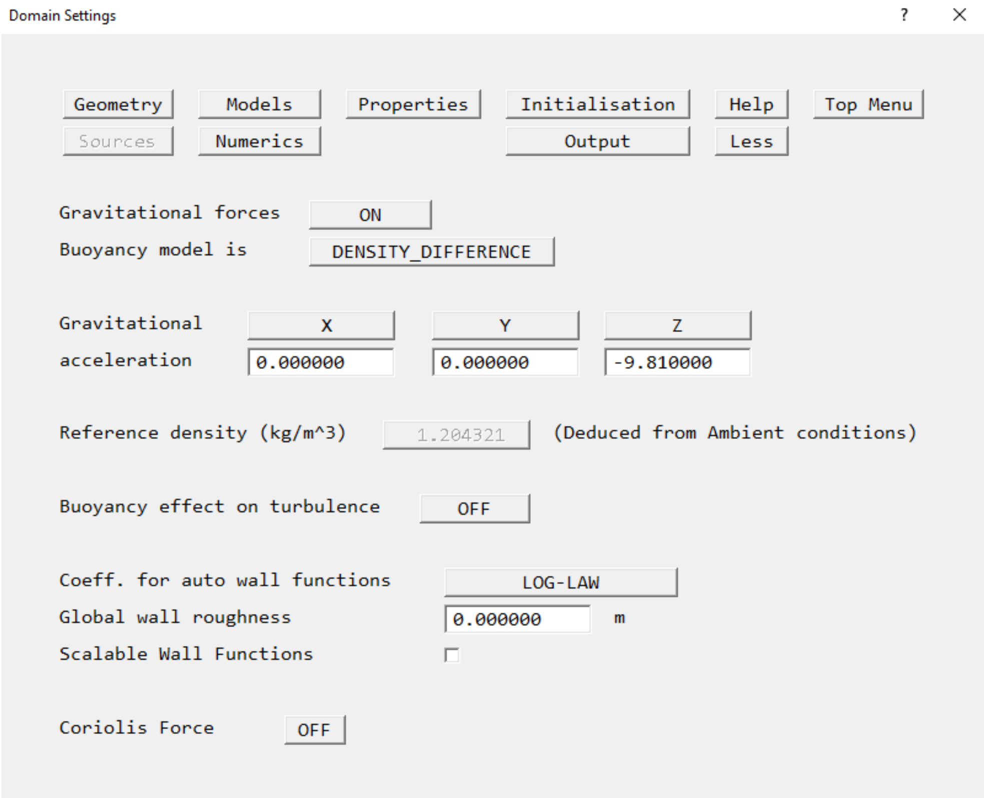


From this panel you can:

- Activate a restart calculation. It is sometimes necessary to perform more iterations than originally set to procure full convergence. Restarting the calculation from the previous solution will save a lot of time.
- If necessary, set individual initial values for each solved variable. This is very rarely needed, as the velocity and turbulence variables are automatically initialised to the wind profile.

8. Sources

The Sources panel (with Energy Equation ON) is shown below.

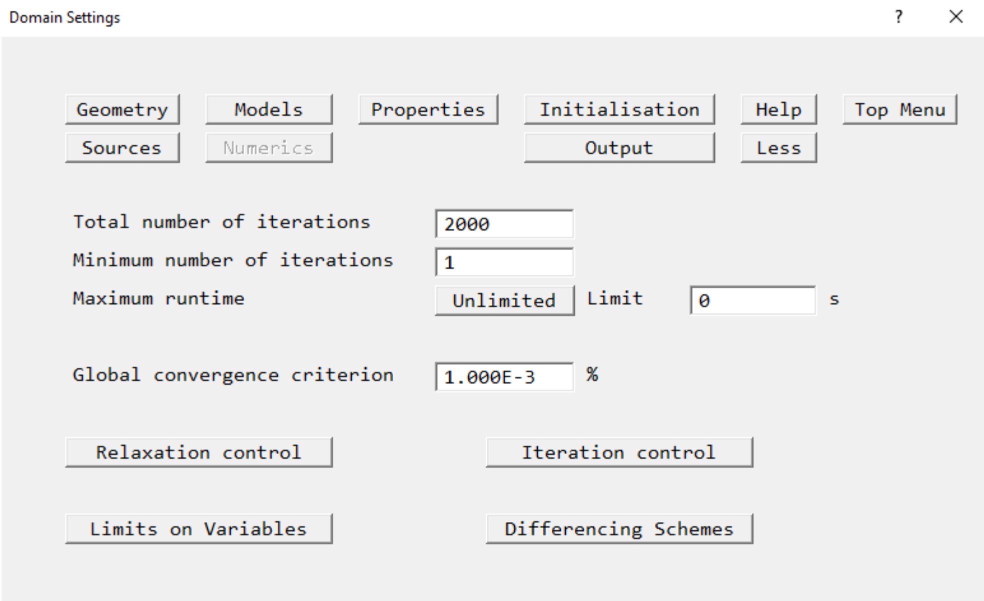


From this panel you can:

- Control buoyancy effects. If the Energy Equation (Section 5.2) is ON, the buoyancy forces are automatically turned on, and the reference density is automatically evaluated from the ambient pressure and temperature set on the Properties panel (Section 6). In practice these settings very rarely need changing.
- Change the direction of the gravity vector. By default, the Z axis points up. If the domain is located on a slope, it can be simpler to change the gravity vector so that it is normal to the ground plane rather than rotate all the geometry.
- Activate the effects of buoyancy on turbulence. This is only likely to be significant if there are strong heat sources, such as fires, present.
- Set the default wall roughness height used for all the buildings. Individual roughness heights can be assigned to individual buildings if required.
- Activate the Coriolis force. This is only likely to be significant of the domain is very large.

9. Numerics

The Numerics panel is shown below.



From this panel you can:

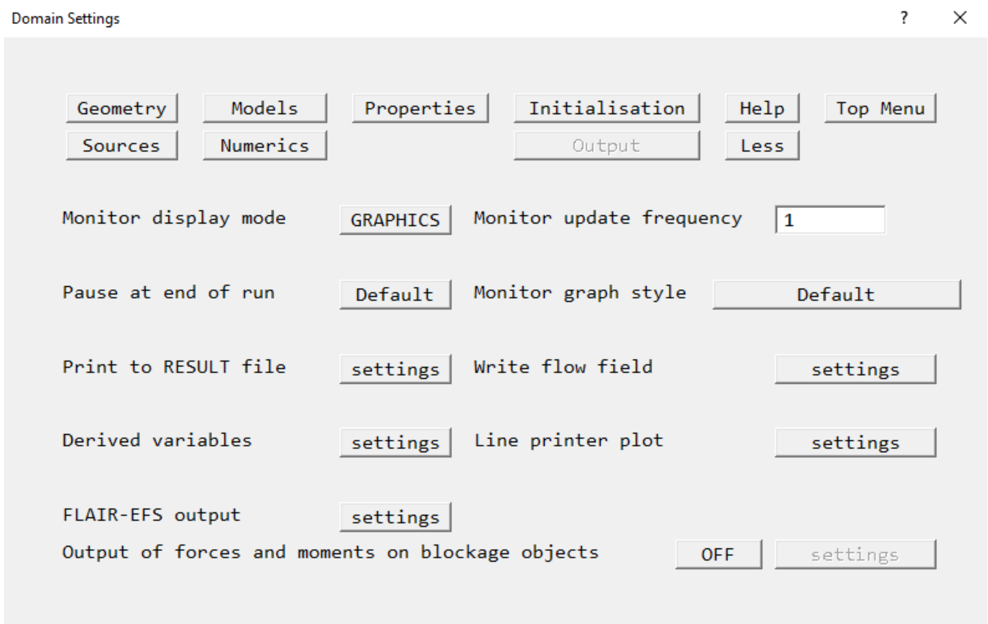
- Set the number of iterations to be performed by the solver. The default 100 is enough to check that the case has been set up correctly, but is usually not enough for convergence. This may require several thousand iterations.
- Access further submenus to control:
    - Relaxation
    - Inner iterations
    - Limits on variables
    - Differencing schemes

These are rarely needed, and are described on-line here.

10. Output

10.1 Output Control

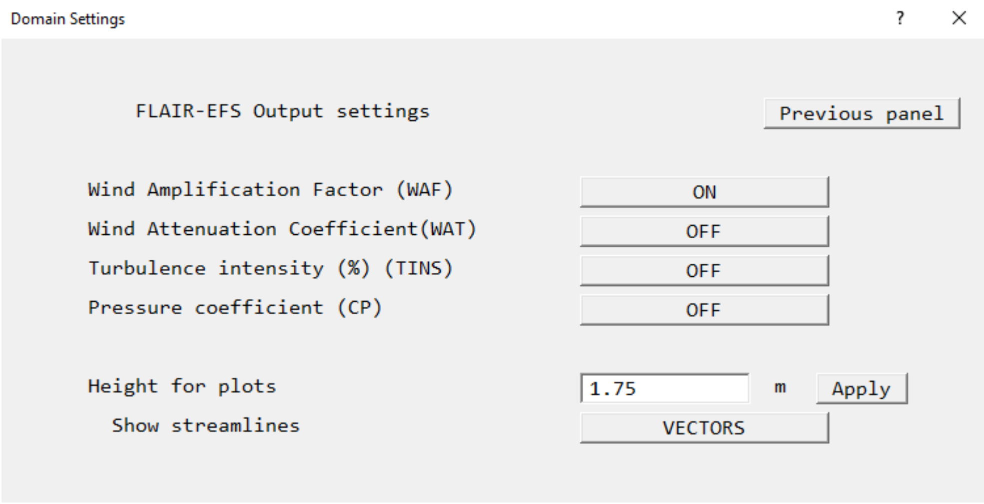
The Output panel is shown below.



This panel gives options to:

- Set the monitoring cell location in terms of cell numbers.
- Set the monitoring cell location in terms of physical space. The nearest cell is chosen as the monitor cell.
- Control the solver convergence monitoring output and monitor update frequency;
- Control the solver end-of-run behaviour.
- Control how the convergence monitoring information is displayed;
- Set field print out controls.
- Select the frequency of field-dumping in terms of sweeps for steady-state cases, or time-steps for transient cases, and select which variables are written to the saved file.
- Activate storage of derived quantities and print-out of wall function information.
- Activate the calculation and printing of forces and moments on objects.

There are also additional FLAIR-EFS outputs available:



The **Wind Amplification Factor** is defined as the local wind speed divided by the wind speed if the obstacle were not present. This is taken to be the wind speed from the wind velocity profile at the local height. The variable name for plotting in the VR-Viewer is **WAF**. This storage can also be activated from the Wind Control Parameters dialog (Section 5.5). For an empty domain, **WAF** would be 1.0 everywhere. It gives an indication of the local increase or decrease in wind speed caused by the building(s).

The **Wind Attenuation Factor** is defined as the **Wind Amplification Factor** minus one. The variable name for plotting in the VR-Viewer is **WAT**. This storage can also be activated from the Wind Control Parameters dialog (Section 5.5). For an empty domain, **WAT** would be 0.0 everywhere. It gives another indication of the local increase or decrease in wind speed caused by the building(s).

The Wind Control Parameters dialog (Section 5.5) also contains options to activate storage of WAF and WAT. It also allows for the storage of a second Wind Amplification Factor (**WAMP**), which is the local wind speed divided by the wind profile speed at a user-set reference height.

The **Turbulence intensity** is a measure of the strength of the local turbulent fluctuations. It is defined as:

$I = 100*k^{1/2}/V_{abs}$

Where k is the turbulent kinetic energy and V_{abs} is the local wind speed. The variable name for plotting in the VR-Viewer is TINS. This storage can also be activated from the Comfort indices panel (Section 5.9.9).

The **Pressure coefficient** is a measure of the force on the buildings it is the local pressure divided by the dynamic head at the wind reference velocity. The variable name for plotting in the VR-Viewer is CP.

10.2 Output Files

The following output files contain useful information:

10.2.1 RESULT

The RESULT file contains:

- An echo of the input parameters,
- Selected numerical values of the solution field (by default 5 * 5 * 5 values are printed regardless of the grid)
- Sums of sources – useful for checking for mass balance, and if the Energy Equation is on, energy balance
- Convergence history.

To view the RESULT file, click ‘**File**’, ‘**Open file for Editing**’, then ‘**Result**’.

10.2.2 INFOROUT

This file contains:

- The maximum value of the Wind Amplification Factor, if it is stored.
- The coordinates of the location.
- The maximum value of Pressure Coefficient, if it is stored.
- The coordinates of the location.
- To view the INFOROUT file, click ‘**File**’, ‘**Open file for Editing**’, then ‘**Inforout**’.

10.2.3 GXMONI

This is a graphics file, usually in .png format, showing the convergence history at the end of the solver run. To view the gxmoni file, click ‘**File**’, ‘**View monitor plot**’, then ‘**Default**’.

10.2.4 PHIDA

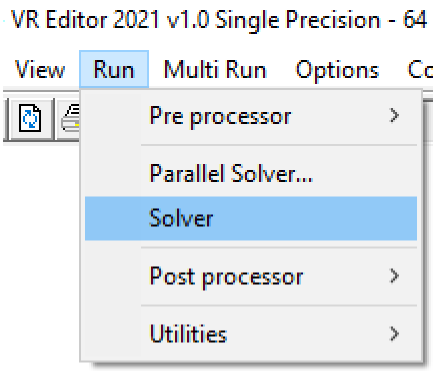
This unformatted file contains the solution fields for all variables in all cells. It is not intended for direct inspection by users, but is used by the Post Processor to display contour and vector plots. It is also used by the Solver when a restart run has been activated.

11. Running the Solver

Once the case has been set up, you will need to run the solver in order to get a solution. The solver can be run in serial (single processor) or parallel (multiple processor) mode. Which to choose depends to some extent on whether your license allows parallel operation, and if it does, how many processors the computer has, and also on how many cells are being used.

11.1 Running the Sequential solver

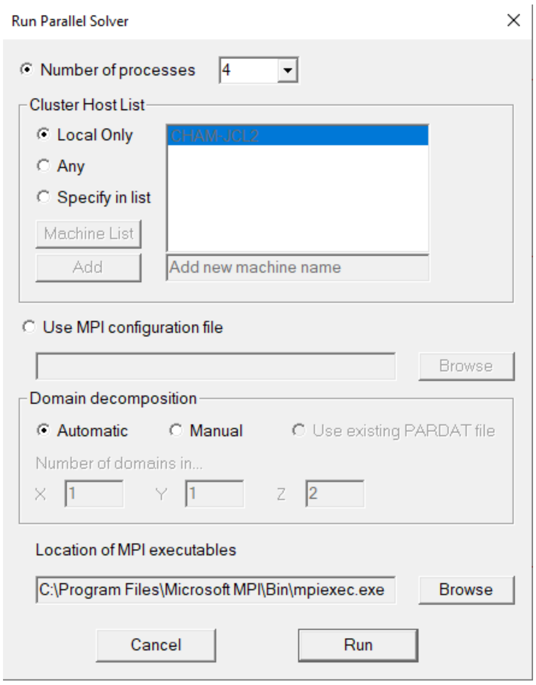
To run in sequential mode, click '**Run**' on the menu bar then '**Solver**'.



On the next dialog (Save Current Settings) click OK and the solver run will commence.

11.2 Running the Parallel Solver

To run the parallel solver, on the above dialog select '**Parallel Solver**' instead of '**Solver**'.



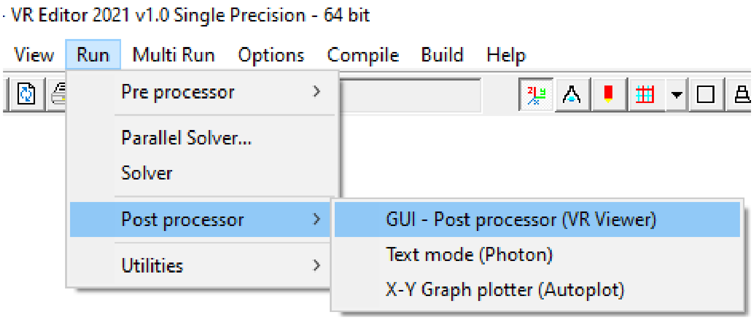
On this dialog select the number of processors to use then click 'OK', and the solver run will commence.

It is not efficient to use more processors than the computer actually possesses, although it is possible. It is also worth remembering that computers with 'hyper threading' will display twice as many processors in the Task Manager as they actually have. For example, a quad-core Xeon processor will show 8 CPUs in the Windows Task Manager. For CFD calculations, only real physical processors count!

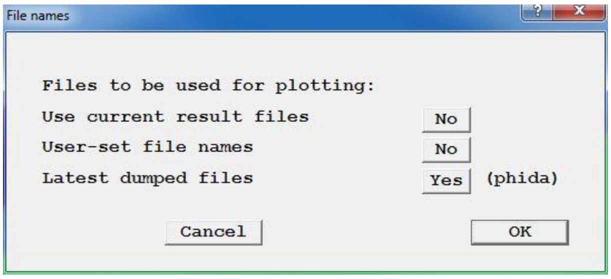
12. Plotting

12.1 Running the Post-Processor

To start the Post-Processor, click Run, Post processor, GUI – Post processor (VR Viewer) on the tool bar.



On the file-selection dialog which follows, click 'OK' to accept the default file names.



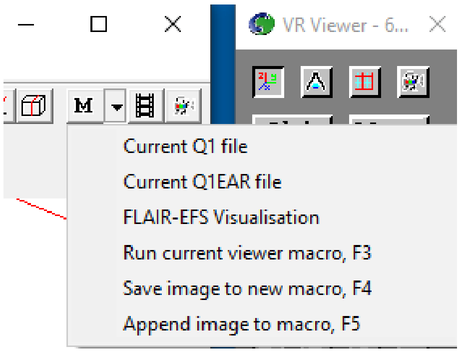
The selected file will be loaded, and the Viewer will start. The Viewer is described on-line here.

12.2 Using the Pre-defined Macro File

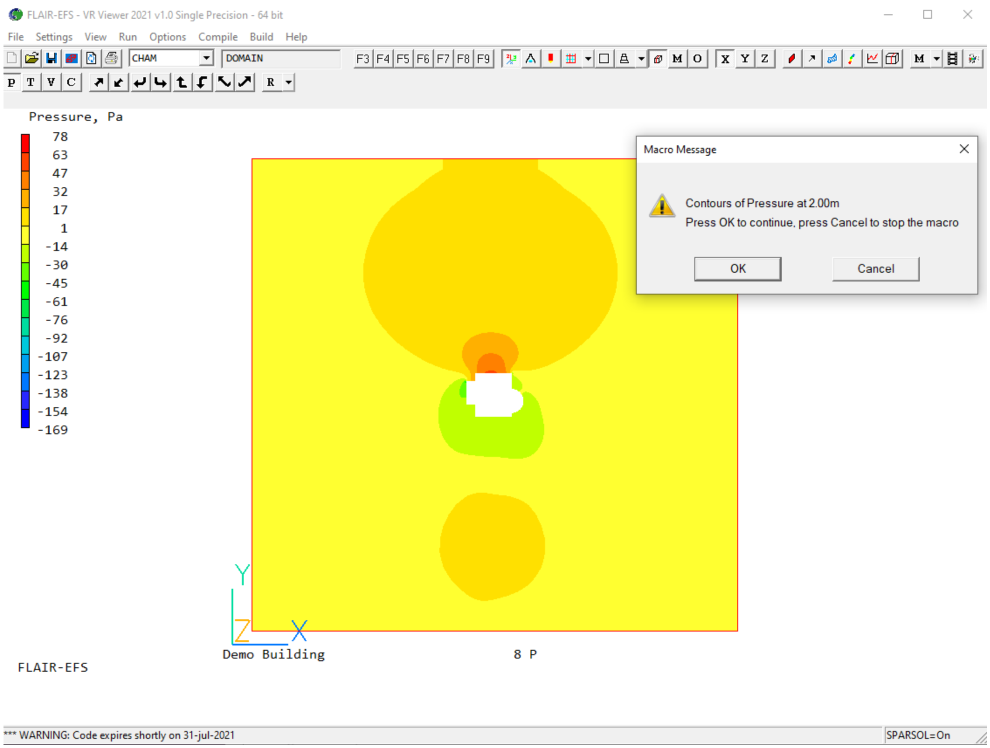
During the pre-processing stage, pre-selected plots can be chosen (Section 10). The instructions for creating these plots are written to a macro file (flair-efs.mac in the local working directory). The full list of plots is:

- Pressure contours on a Z plane at the set height
- Velocity contours on a Z plane at the set height
- Velocity vectors contours on a Z plane at the set height
- Pressure coefficient contours on a Z plane at the set height
- Pressure coefficient contours on the surface of the building(s)
- Contours of Wind Amplification Factor contours on a Z plane at the set height
- Contours of Turbulence Intensity contours on a Z plane at the set height
- Contours of the NEN wind comfort grade contours on a Z plane at the set height
- Contours of the NEN wind danger probability contours on a Z plane at the set height
- Contours of wind speed exceeding the threshold contours on a Z plane at the set height
- Streamlines started at 30 intervals around a circle centred on the region of interest

To run this macro, click the down-arrow next to the macro M toolbar icon, then select FLAIR-EFS Visualisation.



The Viewer will run the macro file. The first image is will be the pressure field at the selected height:



If the ‘Macro Message’ dialog obscures part of the image, you can move it to a more convenient place, and it will stay there. Click OK to advance to the next image in the sequence, or ‘Cancel’ to stop running the macro.

Each image is saved in the current working directory.

The movement controls on the Movement panel can be used when the OK/Cancel dialog is on screen to adjust the view if necessary.

12.3 A brief introduction to the VR-Viewer controls

If you choose not to use the macro, or after the macro has finished or has been cancelled you are free to explore the solution using all the facilities of the Viewer. The Viewer is described on-line here.

To view:

- Velocity vectors - click on the 'Vector toggle' . Vectors are coloured by the current plotting variable, but their length is always related to the absolute velocity.
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Streamline management' button 

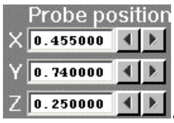
- Iso-surfaces - click on the 'Iso-Surface toggle'

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button'
- To select Temperature - click on the 'Select Temperature button'
- To select any other variable - click on the 'Select a Variable button'

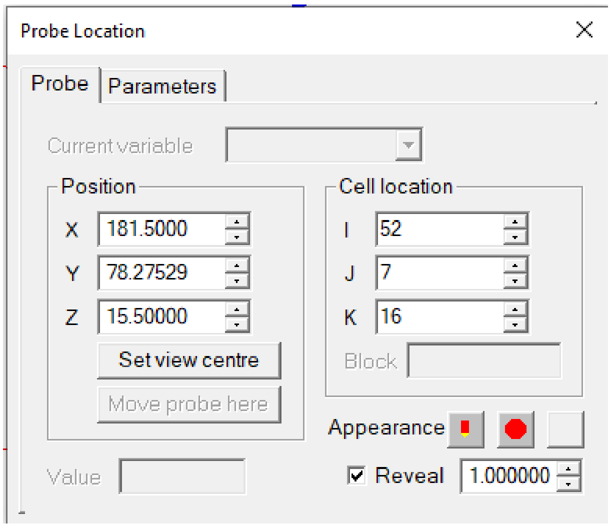
To change the direction of the plotting plane, set the slice direction to X, Y or Z

To change the position of the plotting plane, move the probe using the probe position buttons



.

Alternatively, click on the probe icon on the toolbar or double-click the probe itself to bring up the Probe Location dialog.



13. Case Management

FLAIR-EFS always uses the file named Q1 as the input file. The solver always writes the output files RESULT, PHDA and Inforout. Unless a little care is taken to save the results of one calculation before proceeding to the next, each simulation will overwrite the previous. Some case management tools are provided on the ’File’ menu to assist with this.

The options on the ‘File’ menu allow you to:

- **Start a new case:** everything is reset to the default values. Any previous settings and output files are deleted.
- **Save as a case**: saves all the input and output files using a user-specified base name.
- **Open existing case**: copies all the files linked to a particular case name back to their ‘normal’ names. A full list of files which may be copied is given on-line here.

Users are strongly urged to avail themselves of these possibilities in order to minimise the risk of losing valuable results.

14. References

1) Steadman, R. G. 1973: The Assessment of Sultriness. Part II: Effects of Wind, Extra Radiation and Barometric Pressure on Apparent Temperature. Journal of Applied Meteorology, Vol 18, p874-885

2)https://www.hkgbc.org.hk/eng/beam-plus/beam-plus-references/manuals-assessment/ManualsFiles/BEAMPlus_New_Buildings_v2_0(2021Edition).pdf

Appendix A. Wind Data Formats

This Appendix describes the format of the "wind data" file required for wind averaging (see section 5.9.2).

The first line contains information about the measurements: name, measurement period... This is not used by FLAIR or FLAIR-EFS.

The second line contains the east-west and north-south coordinates of the measurements (not used by FLAIR or FLAIR-EFS) and the **measurement height**.

The third line contains the **number of sectors** into which the complete wind rose is divided (each sector is the centre of a wind direction interval) and two more values not used by FLAIR or FLAIR-EFS.

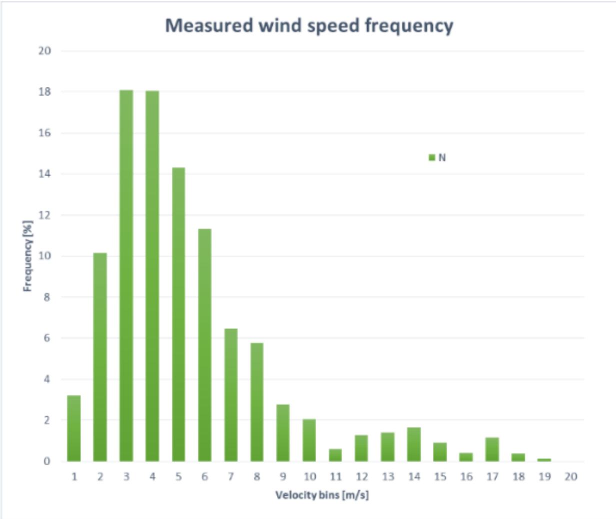
Histogram variant

The fourth line is the **frequency of each wind sector**. If these values are normalised so that they sum to unity, they become the probabilities that the wind is blowing from the respective sectors.

The rest of the file contains values for the **frequencies** for **each wind speed interval** and **wind direction**. From the fifth row onwards, the rows of the frequency table represent wind speed intervals, with each value in the first column being the upper limit of the wind speed interval (in the example below, 0-1 m/s, 1-2 m/s, etc). The columns, from the second column onwards, represent sectors for wind direction. In the example below there are 12 wind sectors; the first sector (commencing "31.99") represents north (i.e. from 345° to 15° in this case), the second N-N-E (i.e. from 15° to 45°).

```
SiteName  01.01.02 - 31.12.02
     65.1   59.9    50.0
      12    1.00    0.00
            3.55    4.97    3.74    4.04    6.70   14.74   17.77   12.67    9.32    9.31    7.38    5.81
   1.00    31.99   22.01   32.63   15.43   16.80    5.38    7.11    7.28   11.49    9.68   18.86   21.01
   2.00   101.37   67.74   56.22   51.83   27.75   19.99   24.21   26.89   40.43   33.86   59.75   76.07
   3.00   180.79  109.88   81.04   66.11   37.48   42.89   41.40   49.91   59.91   43.84   91.14  140.56
   4.00   180.49  134.40   99.31   76.69   41.31   57.08   52.69   59.60   58.07   78.50   94.18  140.38
   5.00   142.89  121.50  106.15   94.15   40.51   62.52   58.46   73.26   69.99   65.30  102.57  121.82
   6.00   113.11   85.80  130.41  105.25   43.86   65.20   67.17   72.58   70.67   81.03  129.91   99.96
   7.00    64.67   94.62  117.28   88.59   57.73   59.62   79.85   81.60   86.37   92.73  104.88   78.46
   8.00    57.76   85.80   92.17   90.18   63.31   73.82   91.32   87.67   85.68   95.94   79.86   54.02
   9.00    27.67   79.13   82.75   77.49   64.43   69.47   88.80   95.68   96.33   96.75   77.11   40.61
  10.00    20.46   72.25   63.92   69.55   69.69   80.05   90.00   84.89   84.07   83.21   67.13   35.10
  11.00     6.02   54.83   44.23   61.62   77.83   86.57   87.90   71.49   72.51   51.87   43.40   43.00
  12.00    12.63   32.47   25.40   51.30   77.51   82.15   72.04   59.18   59.91   49.92   39.93   31.97
  13.00    14.14   26.02   23.40   45.49   73.20   73.17   61.22   57.24   49.25   40.40   23.73   31.24
  14.00    16.54   10.54   18.26   54.21   80.54   56.79   46.38   50.50   51.77   37.30   19.10   22.97
  15.00     9.02    0.86    5.71   23.01   73.52   42.74   38.21   35.41   35.62   33.51   17.36   23.34
  16.00     4.21    0.86    5.71   15.07   53.75   38.03   26.68   28.41   25.66   32.02   12.15   14.33
  17.00    11.43    0.86    5.71   11.11   44.18   23.18   18.93   21.67   17.64   21.58   12.59   10.84
  18.00     3.61    0.43    6.28    1.59   27.43   19.70   17.30   14.67   13.40   19.74    4.63    8.82
  19.00     1.20    0.00    0.57    0.79   16.75   13.76   11.42    8.09    5.96   12.39    1.74    4.04
  20.00     0.00    0.00    2.28    0.53    5.42    9.42    6.25    4.55    2.52    8.26    0.00    1.10
  21.00     0.00    0.00    0.57    0.00    3.83    8.77    4.03    4.05    1.60    5.28    0.00    0.37
  22.00     0.00    0.00    0.00    0.00    1.91    4.20    1.44    2.53    1.15    3.79    0.00    0.00
  23.00     0.00    0.00    0.00    0.00    0.64    3.11    2.28    0.84    0.00    2.18    0.00    0.00
  24.00     0.00    0.00    0.00    0.00    0.32    1.96    1.80    0.67    0.00    0.69    0.00    0.00
  25.00     0.00    0.00    0.00    0.00    0.00    0.29    1.86    0.51    0.00    0.00    0.00    0.00
  26.00     0.00    0.00    0.00    0.00    0.32    0.14    0.90    0.51    0.00    0.23    0.00    0.00
  27.00     0.00    0.00    0.00    0.00    0.00    0.00    0.36    0.34    0.00    0.00    0.00    0.00
  28.00     0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00
  29.00     0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00
  30.00     0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00    0.00
```

The absolute values of the frequencies in this table are not relevant; it is their relative values that are important. After having been read, the frequency values for each wind direction are normalised so that they sum to 1.0, so that the columns of the table become probability density functions for the individual sectors. Thus for example the PDF for the first wind direction can be represented graphically as follows.



The data in the input file can be in free format with a space or ',' as delimiters. Comments and blank lines are not allowed.

Weibull Coefficient variant

The Weibull Coefficient variant has only three lines with wind data. The first one is the **frequency of each wind sector.** The second one is the parameter **A** of the Weibull distribution for each wind sector, and the third is the parameter **k** of the Weibull distribution for each wind sector.

```
SiteName. Distributions specified by Weibull A- and k-parameters
     -55.70         -167.90     12
      12              1.0       0.00
      2.0   4.4   5.6   7.5   6.2   5.3   7.8   8.4   12.2   15.7   16.8   8.1
      3.5   4.4   3.8   4.0   4.3   4.4   5.1   5.9   6.9    6.9    6.3    4.8
      2.10  2.00  1.87  2.62  2.47  2.30  2.19  2.22  2.78   2.38   2.34   1.93
```

The data in the input file can be in free format with a space or ',' as delimiters. Comments and blank lines are not allowed.