

OFWindCFD

User Guide - Version 1.2, 04/2019, corresponding  
OFWindCFD version 2.0.x



IB Fischer CFD+engineering GmbH  
ofwind@cf-d-ing.de

# 1 Inhalt

1	Inhalt.....	2
2	Installation.....	3
2.1	Choose an appropriate computer.....	3
2.1.1	Main memory .....	3
2.1.2	CPU .....	3
2.1.3	Hard disk .....	4
2.2	Installing OFWindCFD.....	4
2.3	Licensing and Activation.....	4
2.4	Make all computer resources available for OFWindCFD.....	8
2.5	Set the correct temp directory to avoid problems .....	9
3	Using OFWindCFD .....	10
3.1	Setting the wind park area and exporting the .flowreq from WindPro.....	10
3.2	Starting OFWindCFD .....	13
3.3	Starting a project .....	13
3.4	Project pane .....	14
3.5	Calculation slider .....	14
3.6	Number of cores slider .....	16
3.7	Direction Pane and Output Window .....	17
3.8	Flowres-Composer .....	25
3.9	How to finish and clean up on ending a CFD project .....	27
4	Troubleshooting.....	27
4.1	Typical problems .....	27
4.2	Temp directory has not enough disk space.....	28
5	Technical Background – What is OFWindCFD? .....	30
6	References .....	35

## 2 Installation

The Installation of OFWindCFD consists of two steps:

- Execute the OFWindCFD.exe application
- Make all computer resources available for OFWindCFD

### 2.1 Choose an appropriate computer

Please be aware that selecting a computer with a fast enough CPU and enough main memory is the most crucial step to achieving acceptable levels of performance with OFWindCFD!

CFD is computationally demanding, so installing OFWindCFD on a too slow machine will definitely reduce the effectiveness of the program a lot!

So, what is currently a sufficiently fast computer?

We have to look mainly at two important hardware requirements:

- 1) The main memory determining the size of the park area and the mesh accuracy that can be set.
- 2) The CPU speed that determines the time you have to wait for results.

#### 2.1.1 Main memory

**Absolute minimum 4 GB RAM** (only run in fast mode, park areas < 22km<sup>2</sup>, your computer will be 100% utilised by OFWindCFD)

**Recommended minimum 8 GB RAM** (good for medium setting in most park areas, accurate for park areas < 25km<sup>2</sup>)

**Recommendation for good performance: 16 GB RAM** or more: No limitations in park area size

#### 2.1.2 CPU

Here the situation is a bit more complex as we need to rely on external CPU speed benchmarks since there are hundreds of different CPU types out there.

So you should check the processor type of your computer by starting the task manager. In the box named "Performance" among others the CPU type is displayed. You can also access the information via the "System" tab of the Control Panel.

With this information you should go to a website called cpubenchmark:

[https://www.cpubenchmark.net/high\\_end\\_cpus.html](https://www.cpubenchmark.net/high_end_cpus.html) and search for your processor

type. Any easy option is to type the CPU's full name into the search menu on the upper right. Please note: even a simple "U" at the end of the CPU's name can make a huge difference (here between low energy and low performance laptop and standard PC version), so be precise in your use of the CPU identifier!

Once your CPU is identified the website yields a CPUbenchmark value (named passmark). It is a 4 or 5 digit number indicating the CPU's speed level.

**A CPU benchmark value above 9.000 (nine thousand) is the acceptable minimum.** With this level you can run a project in fast mode within 24hours (provided you have at least 6GB RAM and do not use the computer for anything else); accurate cases will take about 8 days with this value, large park areas of 30km<sup>2</sup> can take 2 weeks.

**With a CPUbenchmark level of 12.000 (twelve thousand) or more the recommended minimum is achieved:** You can expect to run fast cases over night, normal accurate projects within 5-6 days, large ones (30km<sup>2</sup> park area) within 9-10 days.

**If a new computer is bought for OFWindCFD** we recommend a CPUbenchmark level of 17.000 minimum (seventeen thousand), typically found on a CPU with 12, 16 or even more cores. Combine this with at least 16GB RAM, preferably 32 GB for maximum performance.

### 2.1.3 Hard disk

CFD produces a lot of data and especially if it is kept for later, the hard disk can fill up quickly (particularly if you have a standalone PC and not a file sharing system). A hard disk capacity of a couple of terabytes (2-6 TB) is recommended. Smaller hard disks do not affect calculation speed, they just make it necessary to clean the disk more often to avoid calculations stopping due to lack of disk space.

## 2.2 Installing OFWindCFD

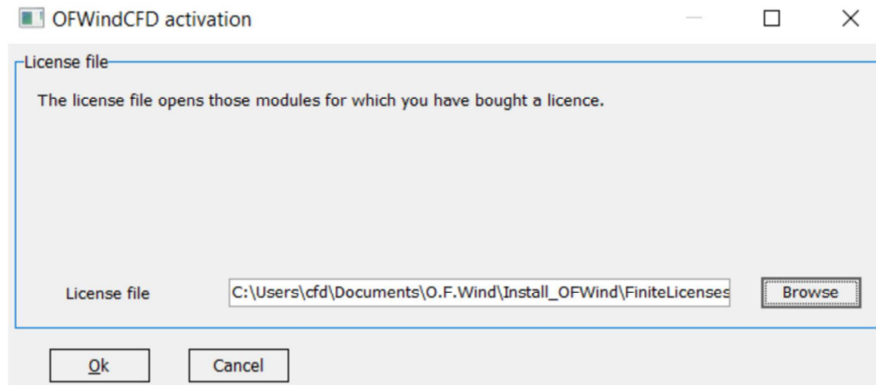
Please make sure that your computer has a 64-bit version of Windows 7 or higher. When purchasing OFWindCFD you have received a file called "OFWindCFD-Setup\_xxx.exe". Double click on it and follow the instructions. If you haven't installed MPI version 8.1 please confirm the installation of MPI as OFWindCFD needs this version to work properly. When OFWindCFD starts for the first time you have to register your license.

## 2.3 Licensing and Activation

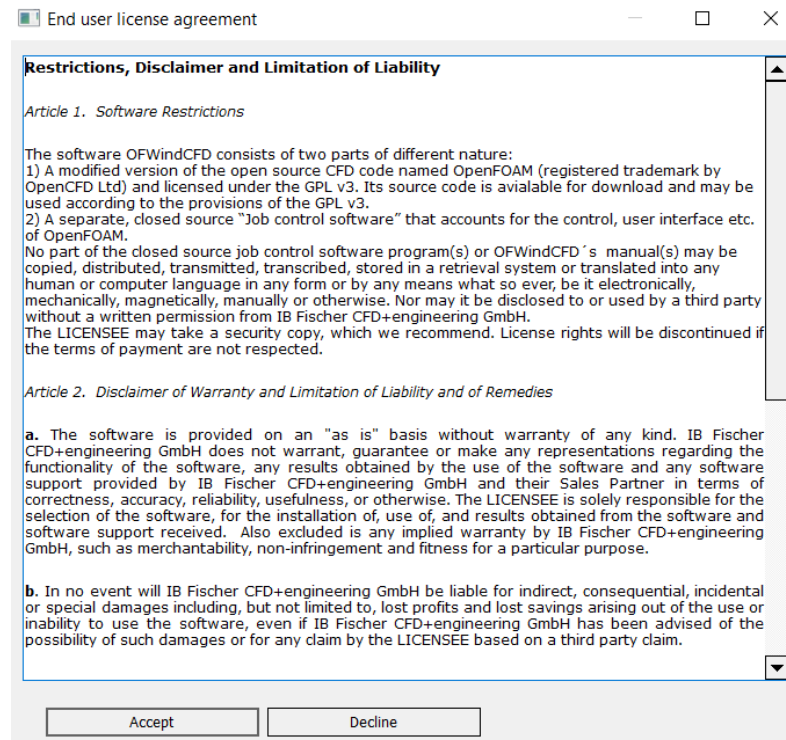
A license file is emailed by EMD to OFWindCFD clients. Please contact EMD, if you change your e-mail address or have not received it. License files are by default e-

mailed to the primary contact EMD has on record for each company. If you are in any doubt as to the primary contact details, please refer to EMD for information.

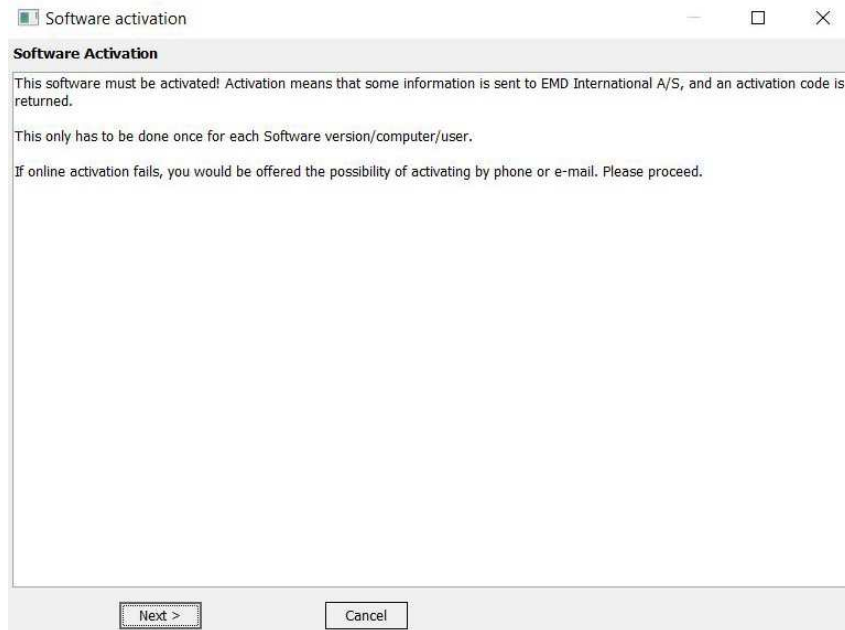
When starting up OFWindCFD for the first time, the program will automatically ask for a license file (see image below).



Navigate on your computer to the license file you have received and confirm by clicking "OK".



Accept the license agreement and continue to the next window.

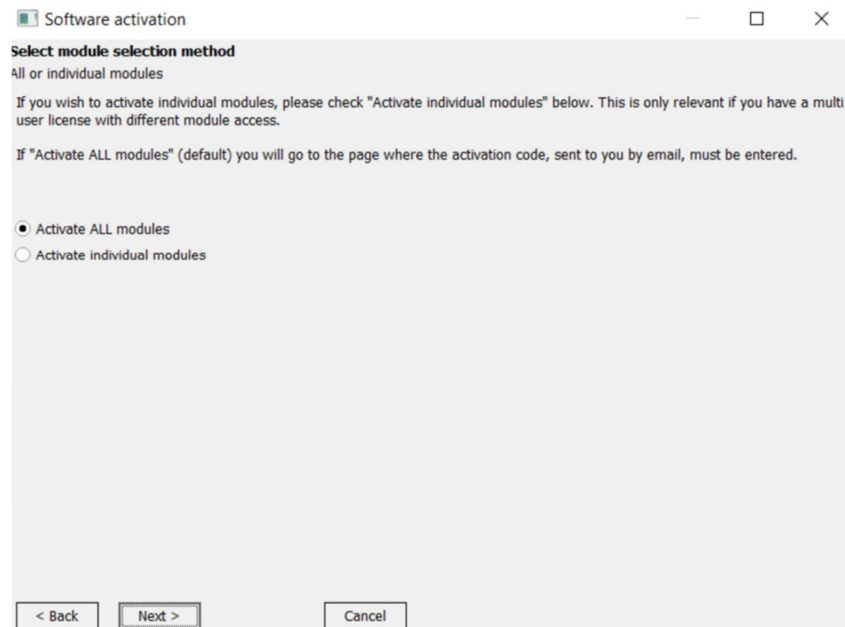


Here click on "Next" and enter your name and e-mail address in the text boxes.

The image shows the "Software activation" window at the "Activation information" step. It contains several input fields and options:

- User information:**
  - Application Id: OFWindCFD 2.0 (dropdown)
  - Disc Id: TS351937
  - Computer Id: CFDLAP2
  - User name: cfd
  - Customer Data: 6652, IB Fischer CFD+engineering GmbH, Lipowskystr. 12
- I am:**
  - ☒ End user
  - ☐ Administrator
- End user:**
  - Name: YOUR NAME
  - E-mail: YOUR E-Mail ADDRESS
  - Repeat e-mail: YOUR E-Mail ADDRESS
- Instructions:** "This information will be sent to the activation server. Press Next to start the communication." and "FIREWALL ISSUES: You might temporarily need to allow the program to communicate with the EMD server using HTTP."
- Option:** ☐ Skip online activation, and activate by phone or e-mail
- Buttons:** "< Back", "Next >", and "Cancel"

When clicking “Next” you get to the following window in which you select whether you want to activate all modules or individual modules. Select your choice and click “Next”.



Software activation

**Select module selection method**

All or individual modules

If you wish to activate individual modules, please check "Activate individual modules" below. This is only relevant if you have a multi user license with different module access.

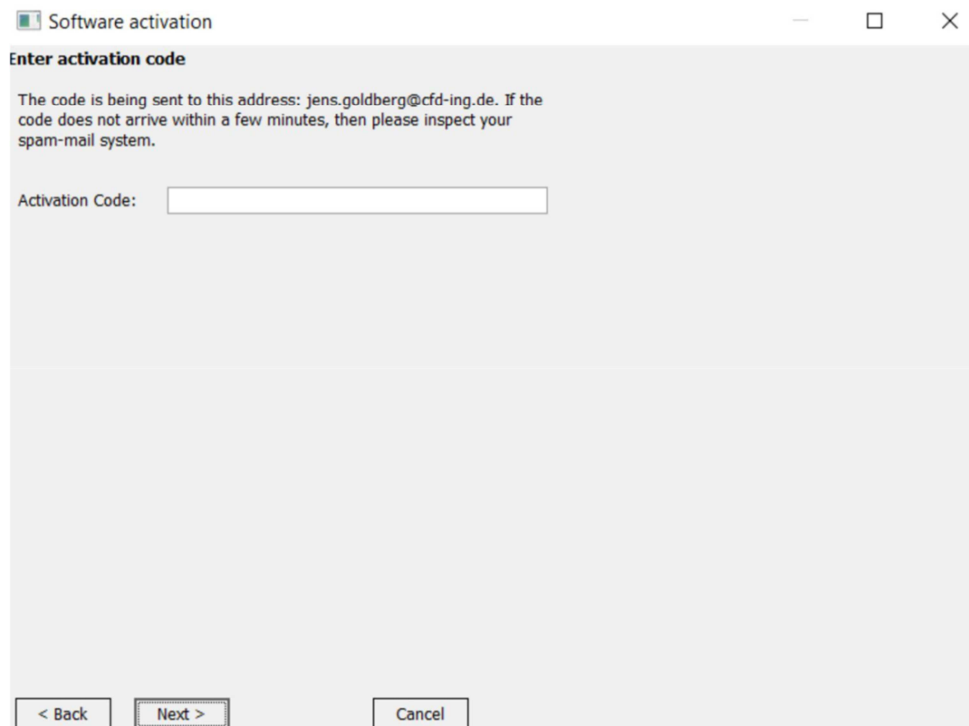
If "Activate ALL modules" (default) you will go to the page where the activation code, sent to you by email, must be entered.

☒ Activate ALL modules

☐ Activate individual modules

< Back   Next >   Cancel

You will then receive an e-mail by support@emd.dk which includes your activation code for OFWindCFD. Enter this code into the prompt in the next window and the activation is completed.



Software activation

**Enter activation code**

The code is being sent to this address: jens.goldberg@cf-d-ing.de. If the code does not arrive within a few minutes, then please inspect your spam-mail system.

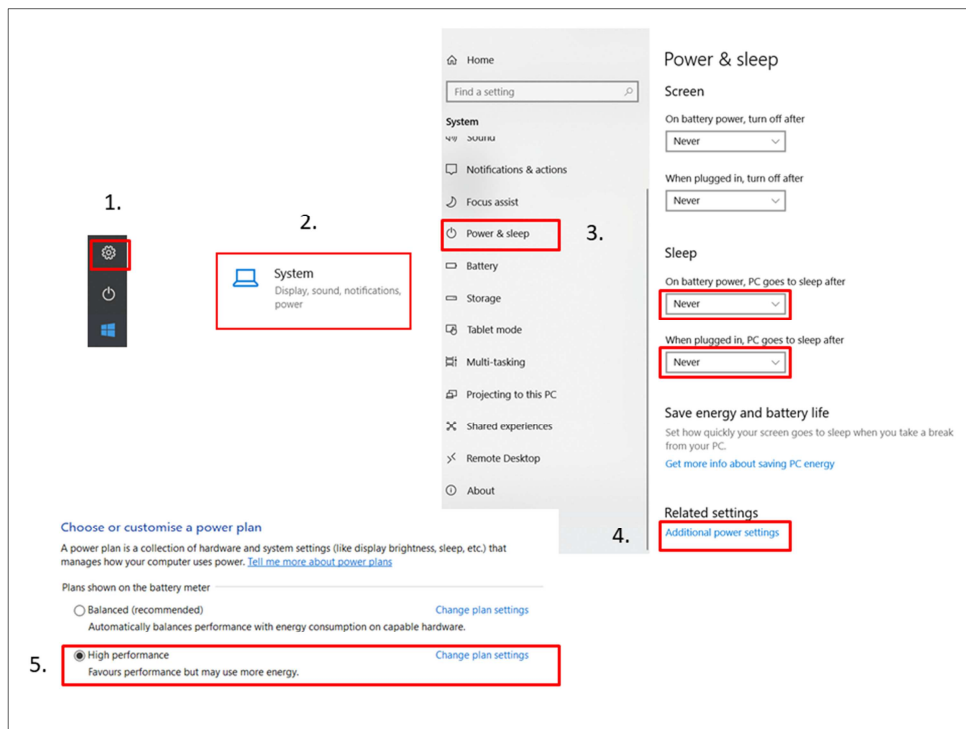
Activation Code:

< Back   Next >   Cancel

## 2.4 Make all computer resources available for OFWindCFD

By default Windows is configured in a way that the energy is saved. As the simulations in OFWindCFD are quite complex the computation time will be dramatically higher if not all hardware is used that is available. To make sure your computer lets OFWindCFD use all resources follow these instructions:

1. Click on the Windows symbol and select the „Settings“ icon.
2. In the next window select “System”.
3. In the following window select the “Energy-Saving” tab and set all switches for standby mode to „Never“
4. Click on the „additional energy saving options“ link on the bottom
5. Here select the „High Power settings”





## 2.5 Set the correct temp directory to avoid problems

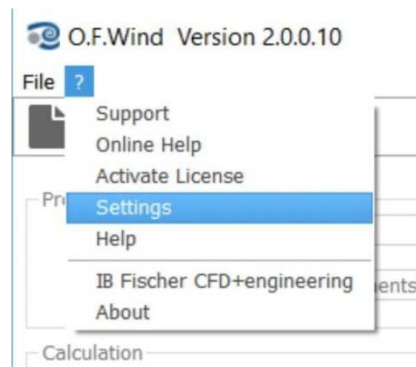
The directory where all data are stored during the simulations is the OFWindCFD working directory which by default is created in USER/AppData/Local/Temp/ (in Windows 10).

**This temporary folder may run out of space and hence stop the computational run.**

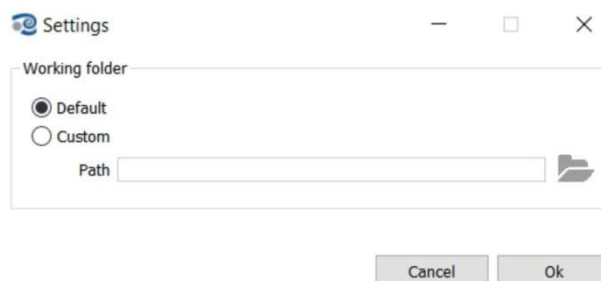
**ATTENTION: In major Windows 10 updates the User/.../Temp directory is cleaned! So, if you save important data in this folder it will be lost. Please backup somewhere or choose a working directory different from USER/.../Temp**

**It is strongly recommended to change the default temp directory to a different place chosen by you on your file system.**

In order to change the temp directory to a different place choose the “?” icon at the top and click on “Settings”



In the next window change the selection to “Custom” and click on the folder icon. Then you can navigate to the directory of your choice and select it.



(See more details in chapter 3.3 if necessary)  
Having done this, the OFWindCFD installation is completed.

### 3 Using OFWindCFD

In order to use OFWindCFD, a flowrequest archive is needed to import the map and roughness data into the simulation process.

#### 3.1 Setting the wind park area and exporting the .flowreq from WindPro

These are the steps for creating a flowrequest-archive in WindPro:

- 1) Height and roughness maps for the site must already be loaded (e.g. by creating line objects).
- 2) Then a terrain data object can be created which refers to the line objects.
- 3) The most important step is to define a polygon as “WTG Area” Object.

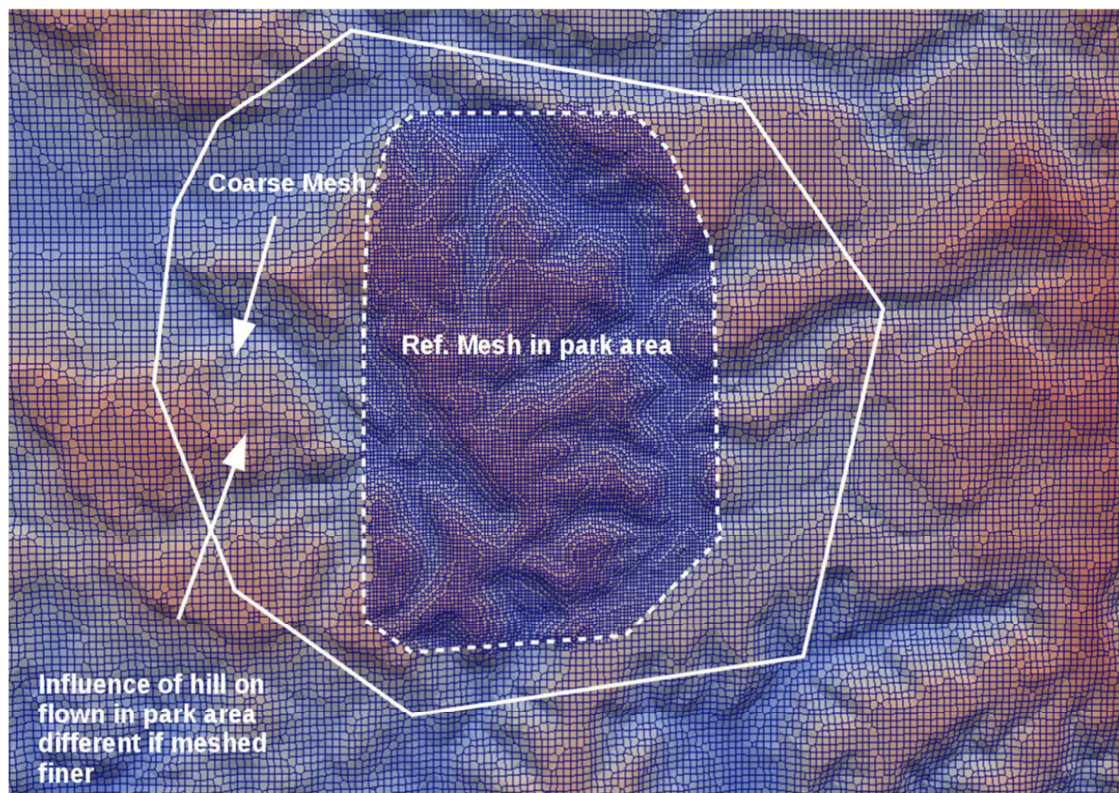
This polygon shall cover at least the whole windpark area. It also defines the area of highest mesh resolution in the CFD model.

**This “Park Area” will also be the output area in which you will see the flow field parameters and be able to make calculations within.** As a consequence you may want to define:

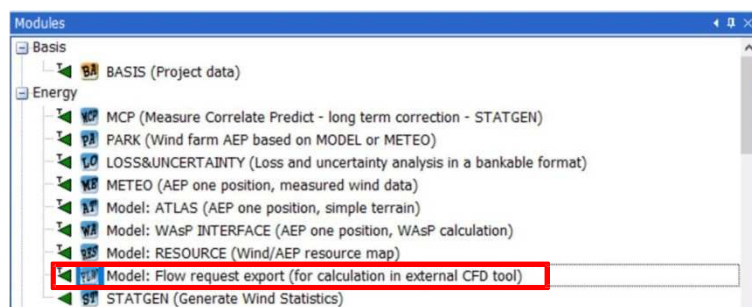
- Small Area around turbines for energy yield only
- Bigger Area for further flow field investigations
- Minimum Distance from turbines or met mast to park area edges should not be below 500 m (better: 1000m)
- Keep the park area object in a simple shape (rectangular, simple polygon) with not too many points (this avoids potential trouble at later stages)
- Inspect the borders of your park area over the terrain model to be sure to include/exclude terrain features in a useful way: If a steep hill or other distinct orographic features are just outside the real wind park area it is a good practice to include them into the CFD park area since mesh resolution is finest here and complex orography is resolved better with a fine mesh. In case of doubt make the park area rather bigger than smaller.
- Looking at the CFD results one more thing has to be considered as a consequence of the above: If the park area borders are moved over distinct orographic features from one park development stage to another, small changes in results cannot be excluded. The reason for that is: Flow features are re- solved slightly different in a coarse mesh (i.e. small park area) compared to a fine mesh (big park area). - See also Image on next page:

- ☒ Scenario 1: The park area (=fine mesh area) is just the inner polygon (dotted line)
- ☒ Scenario 2: The park area is the outer polygon (full line)

So expanding the park area from the small / dotted to the bigger / full line polygon will result in a small change in results: In scenario 1 (small area) the hills on the left are resolved with a coarser mesh compared to a bigger park area (full line) where the hills are resolved with a fine mesh – and such is the flow behind the hills, which will result in a slightly different flow profile in the park area.



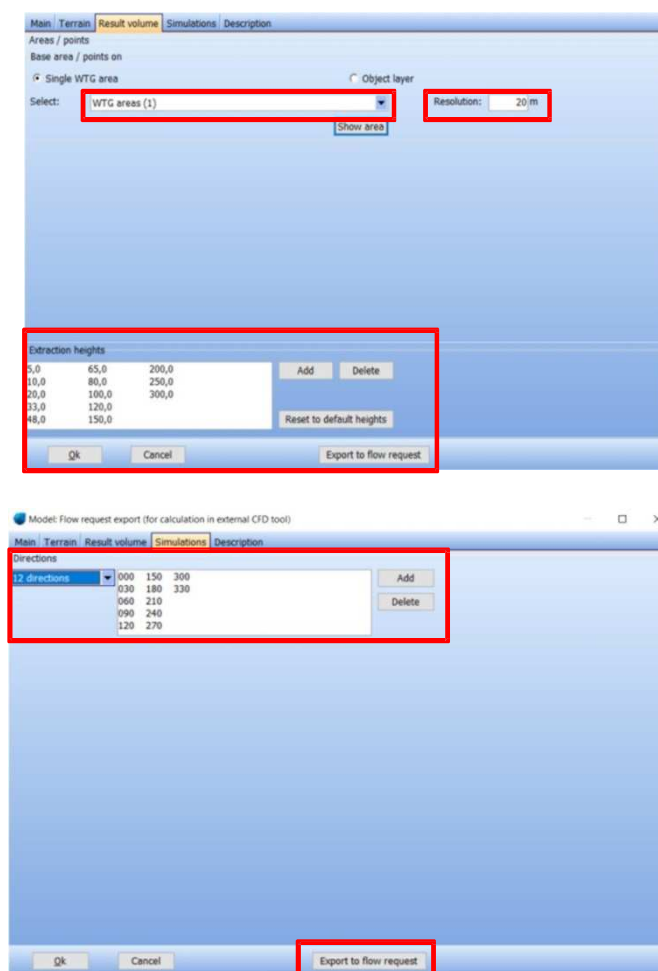
4) In the modules selection window then double click on “Model: Flow request export” (see figure below).



5) In the “Result volume” tab the user selects the WTG-Area for which the simulations should be conducted. The resolution value defines on how many points (leading to a finer or coarser resolution) the data inside the WTG-Area is displayed after the simulations are finished.

**Please note: This step only defines the display settings in WindPro, it does NOT affect the definition of the computational grid (i.e. mesh resolution) in OFWindCFD itself! The mesh resolution is defined within OFWindCFD at a later stage.**

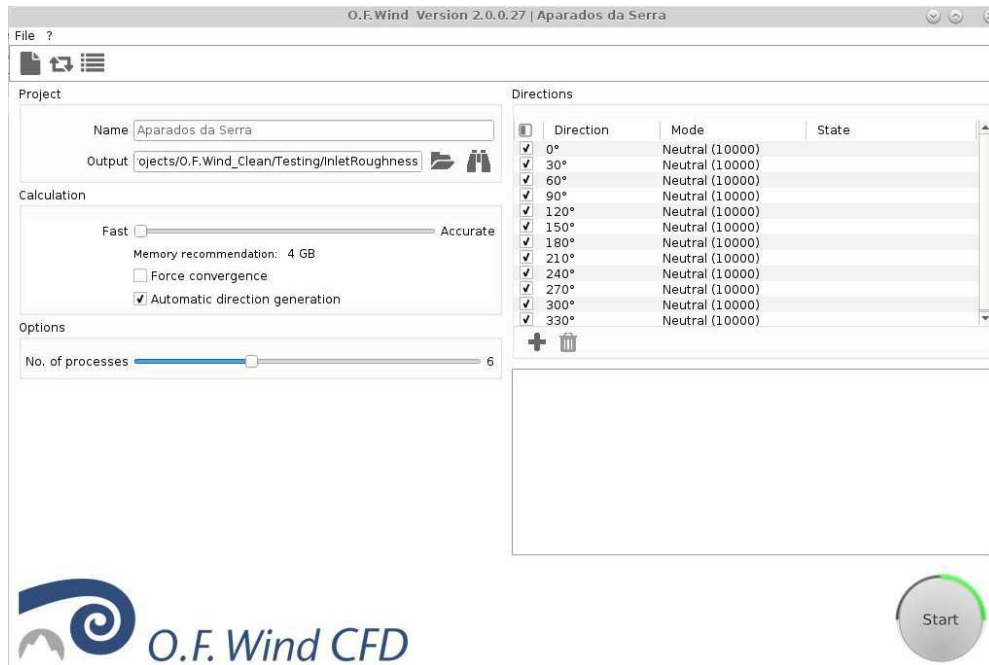
The heights the user enters in the bottom window are the heights above ground level in which the simulation results will be shown. In the next tab “Simulations” the user can add or remove wind directions which should be simulated. (This definition of wind directions can also be made later in OFWindCFD itself)



For finishing the user should click on “Export to flow request” to save the map, roughness and simulation request data to an archive called “...flowreq”.

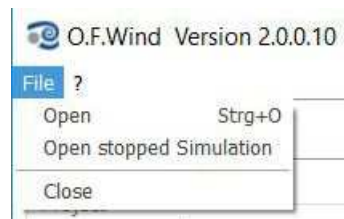
### 3.2 Starting OFWindCFD

After installing the software you can double click on the OFWindCFD logo to start the program. On the next page is the OFWindCFD Window shown which then opens up with its five main sections. These are described on the next pages.



### 3.3 Starting a project

To start a project click on "File" and select "Open":

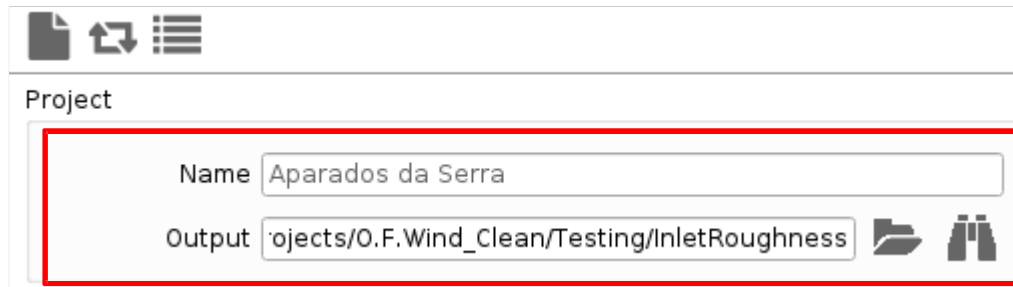


Then navigate to the destination where you saved the flowreq-Archive from WindPro and load it into OFWindCFD.



### 3.4 Project pane

In the project Pane the name of the site is shown, according to the corresponding windPRO project. The “Output” field shows the path where the user has chosen to save the simulation results file, the so called flowres-Archive. If you start OFWindCFD for the first time this field is empty.



By clicking on the binoculars a Windows Explorer window opens up to look for a place where simulation data is already stored or where free disk space is available. To set the path, click on the folder icon to navigate to a place of your choice.

### 3.5 Calculation slider

#### Quick theory behind the CFD calculations:

OFWindCFD uses the OpenFOAM® libraries and applications which are based on the Navier-Stokes equations. Basically, these equations describe the conservation of mass, momentum and energy inside a flow field. To use the Navier-Stokes equations a computational grid is needed to fill the volume which should be computed. In our case this is the WTG-Area and the surroundings as well as the vertical space above the terrain.

For more information on theory please refer to chapter 4 of this user guide.

#### The practical consequences of this:

**Accuracy as well as computation time heavily depend on this computational grid consisting of many millions of cells:** The finer the grid resolution is set, the more cells are needed to describe the volume and the more accurate the solution gets. However, the computation time also increases with the number of cells. That is why both targets - small computation time and accurate solution - can't be met at the same time.

A second and similar factor is the number of wind directions that are computed: The more directions you run – especially in the site's predominant wind directions – the more accurate the solution gets.

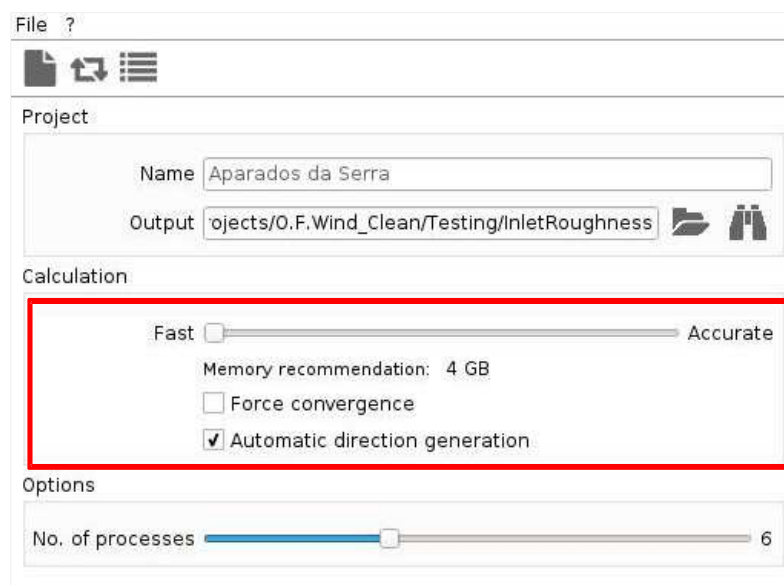
## Your choice: Fast – Medium – Accurate

You can choose between three different settings that determine OFWindCFD's speed and accuracy: Fast – Medium – Accurate. These three positions decide which spatial resolution is used inside the WTG-Area of the computational grid – and the number of wind directions to be run.

**“Fast”** means: A spatial resolution of 31m and 12 directions pre-set (30° sectors)

**“Medium”** is: A spatial resolution of 26m and 18 directions pre-set (20° sectors)

**“Accurate”** is: A spatial resolution of 23m and 36 directions pre-set (10° sectors)



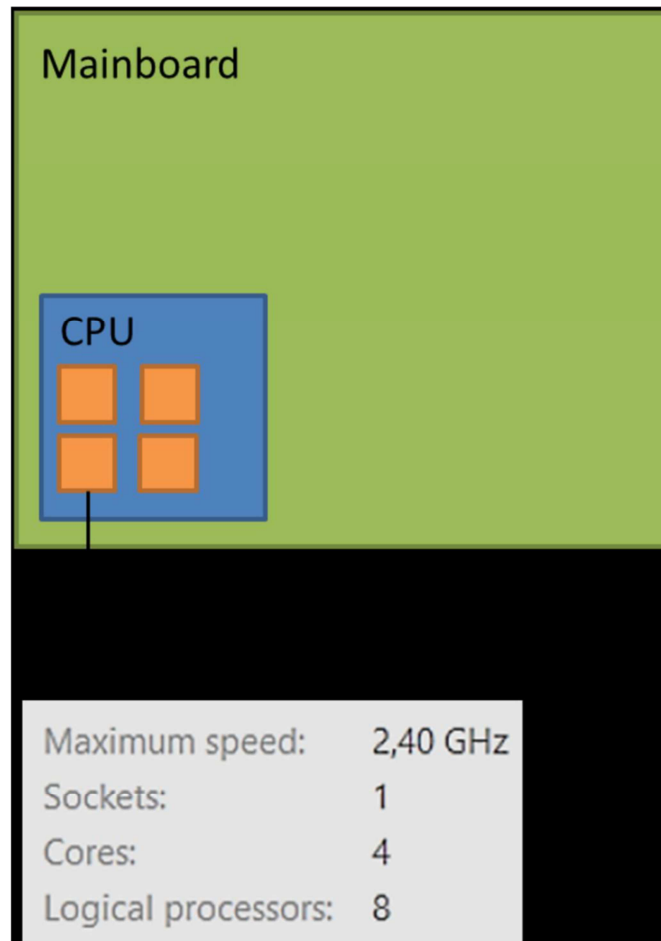
With the number of cells increases the required working memory (RAM) for the simulations - so it can happen that the memory of your computer is not enough. Hence a warning comes up when OFWindCFD recognizes not enough memory. In this case change to a computer with more RAM or set the slider to “Fast”.

In some cases, it happens that the standard schemes to solve the discretization of the Navier-Stokes Equations on the grid doesn't lead to stable solutions. For these cases OFWindCFD has the check box “Force Convergence”. If a user experiences such a case this check box can be selected and the simulations are started again.

By deselecting the box “Automatic direction generation” the number of wind directions to simulate is not changed with changing the slider.

### 3.6 Number of cores slider

A short explanation on the wording is necessary here. The hardware of your computer includes one or more CPUs like Intel i7, AMD Opteron etc. A CPU (central processing unit) can have multiple numbers of cores, also named processors. The image below shows an example how the hardware can look like.



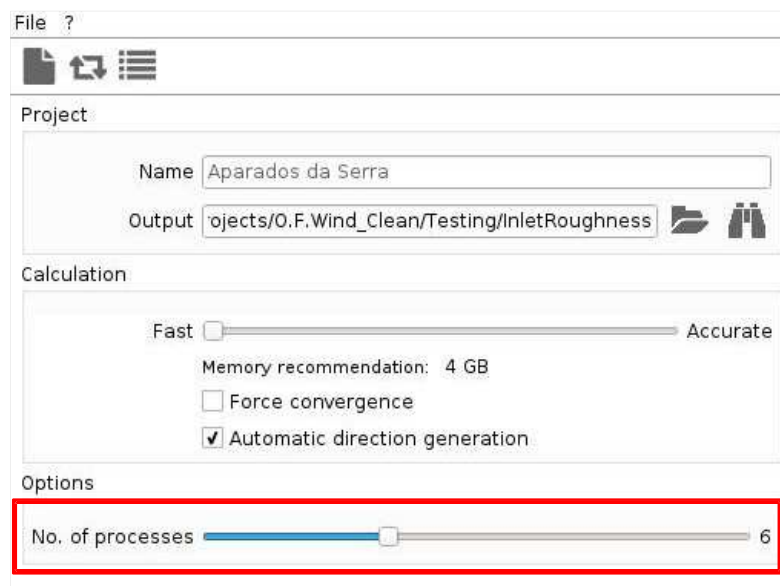
If you have a look at the system information when starting the task manager, you can find the number of your computer's cores on the second entry under "Performance". The example here has one CPU and four cores/processors. The number of logical processors is always double the number of physical cores. However, this doesn't have a physical meaning but refers to a special administration technique of tasks (hyper- threading) to optimise parallel processing. Bear in mind that the number of processes selected in OFWindCFD should not be higher than the number of physical cores your computer has!

In the "Number of processes" pane the user can select how many processors of the computer should be used for the simulation. What OFWindCFD then will do is to



parallelize the simulations on the number of processes the user has selected. The cores then receive the number of started processes to work on them. There will be two different licenses available:

- ☒ OFWindCFD Basic: Parallelisation available from 1 to 6 processors
- ☒ Extra Package for more Cores: Parallelisation available from 1 to 16 processors



**The number of processes selected in OFWindCFD should not be higher than the number of actual cores your computer has!** If a user selects more processes than processors this can lead to a state in which the computer doesn't respond until the simulations are finished or gets stuck in the middle of the simulation.

### 3.7 Direction Pane and Output Window

In the pane with the scroll bar you can see the wind directions pre-set by OFWindCFD according to the calculation slider (fast 12 – medium 18 – accurate 36 directions). You can add or delete directions with the “+” and “trash can” symbol on the bottom. It is recommended to have a 10° stepping of directions at least in the site's predominant wind directions.

**Having selected the number of cores it is time to start the simulations by clicking on the “Start” button.**

After pressing the “Start” button, the column “State” is updated for each direction. There are four different states:

- Waiting: Every direction which has not been simulated yet is in this state
- Running: The wind direction currently being simulated
- Successful: After a direction has been simulated it gets the state “Successful”
- Failure: If an error occurs *or if the user stops the run of a direction* the state changes to “Failure”. Then this direction has to be computed again at a later time.

Information of the ongoing simulation run is also being printed out into the pane below the direction list.

The screenshot displays a software interface for wind direction simulation. At the top, a table titled "Directions" lists 12 directions from 0° to 330° in 30° increments. Each row includes a checkbox, the direction angle, the mode "Neutral (10000)", and the current state. A red rectangle highlights the first four rows (0° to 90°). Below the table are icons for adding (+) and deleting (trash) directions. A log window below shows a series of timestamped messages detailing the simulation process, including mesh generation, probe application, case setup, and solving for different directions. At the bottom right is a large circular "Stop" button with a red progress indicator.

	Direction	Mode	State
<input checked="" type="checkbox"/>	0°	Neutral (10000)	Successful
<input checked="" type="checkbox"/>	30°	Neutral (10000)	Failure
<input checked="" type="checkbox"/>	60°	Neutral (10000)	In progress
<input checked="" type="checkbox"/>	90°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	120°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	150°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	180°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	210°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	240°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	270°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	300°	Neutral (10000)	Waiting
<input checked="" type="checkbox"/>	330°	Neutral (10000)	Waiting

Log messages:

```
2019-03-27 13:03:33 generating CFD mesh...
2019-03-27 13:08:06 generating probes...
2019-03-27 13:08:10 applying roughness...
2019-03-27 13:09:59 reconstructing parallel mesh...
2019-03-27 13:13:26 setting up case 000 MOL 10000
2019-03-27 13:13:27 solving case 000 MOL 10000
2019-03-27 14:17:16 sampling results...
2019-03-27 14:24:00 generating flowres files...
2019-03-27 14:24:15 setting up case 030 MOL 10000
2019-03-27 14:24:16 solving case 030 MOL 10000
2019-03-27 14:45:53 sampling results...
2019-03-27 14:45:53 generating flowres files...
2019-03-27 14:45:53 setting up case 060 MOL 10000
2019-03-27 14:45:54 solving case 060 MOL 10000
```

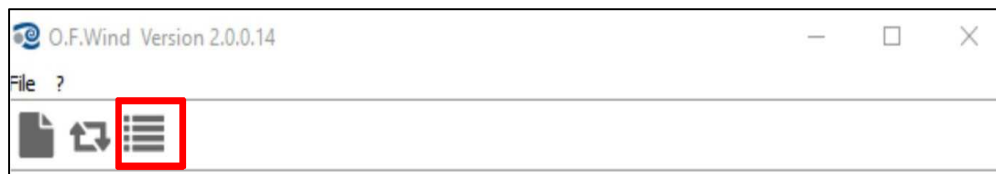
Stop

### 3.8 Flowres-Composer

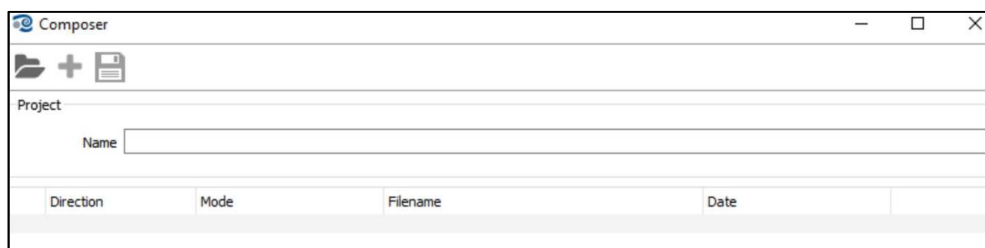
When working with OFWindCFD the situation can come up, that you want to join simulation data from two or more different simulation runs. Let's say you want to have a quick rough estimate of where the highest speed ups are in a WTG-Area and start a project with 0°, 30°, 60°... (12 directions). The simulations take 12 hours as your computer is quite powerful, the WTG-Area is not that large and you choose "FAST" simulation settings. Hence, the next day you have your results in a flowres-Archive. Now you want to simulate the other 24 directions (10°, 20°, 40°, 50°...) and start a second project on the same flowreq file while working with the first rough estimate. After these simulations are also finished you have a second flowres-archive. With the Flowres-Composer you can now merge both result-archives together so you can import all simulation information into windPRO at once.

To start the Composer, click on the right icon:

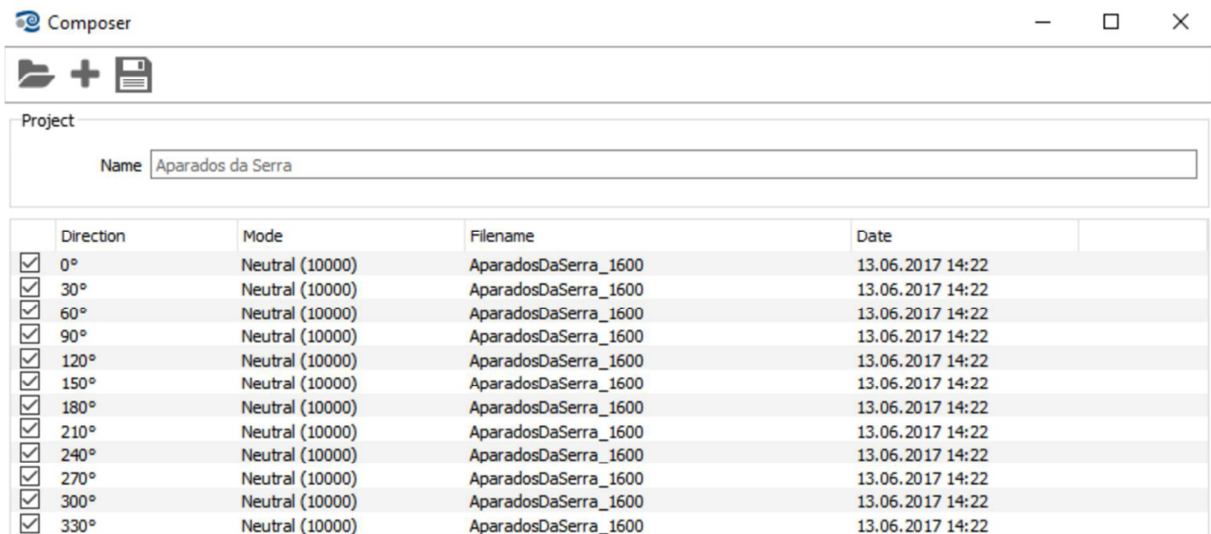
The window which then opens up shows you three icons:



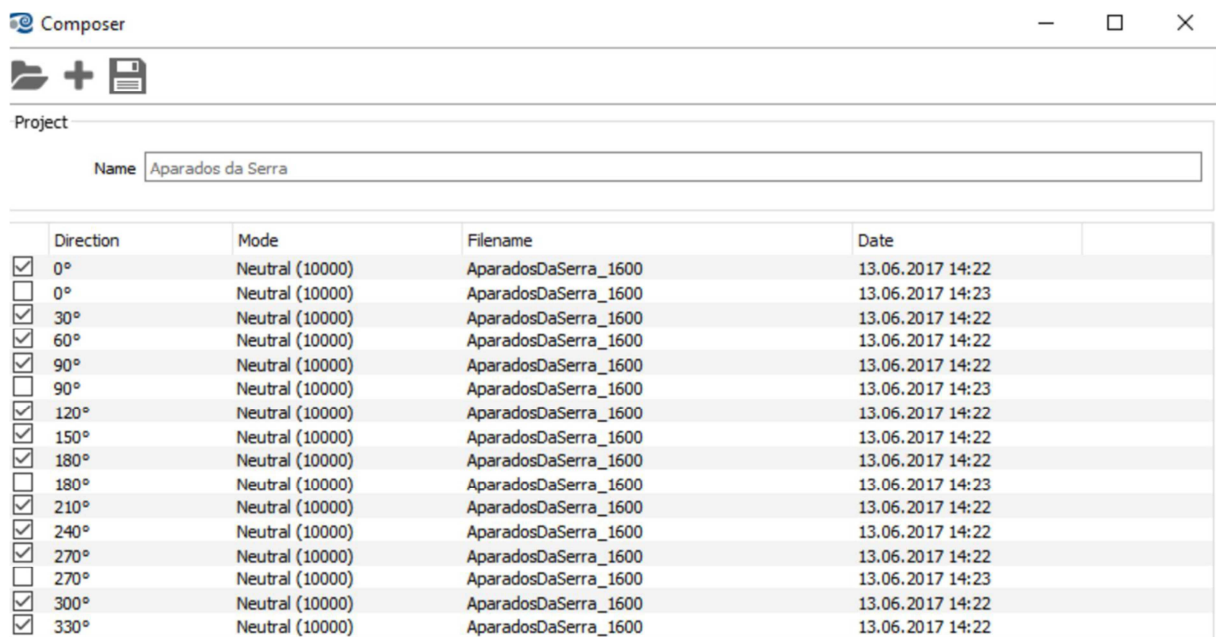
Open flowres-archive – Import flowres-archive – Save your selected directions to a new flowres-archive



To start the process, click on the “Folder-icon” and select a flowres-archive you want to start from. After it has been opened the pane should look something like this:



Then click on the “Plus-icon” and select an additional archive:

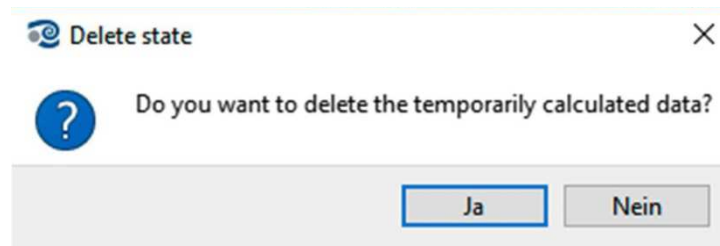


As you can see, the composer automatically deselects all directions which are already calculated. This is a precaution as in the flowres-archive each wind direction is only allowed to be present one time. To finish the process, select the “Disk-icon” and save the selected directions to a new flowres-archive.

### 3.9 How to finish and clean up on ending a CFD project

After closing OFWindCFD, the program asks if temporary calculation data should be deleted. On clicking yes just the .flowres files are stored, which are the input for windPRO. All other space-consuming data are deleted.

You should choose to delete temporarily calculated data. There is only one exception: Do not delete, if data are needed for debug/support reasons.



## 4 Troubleshooting

### 4.1 Typical problems

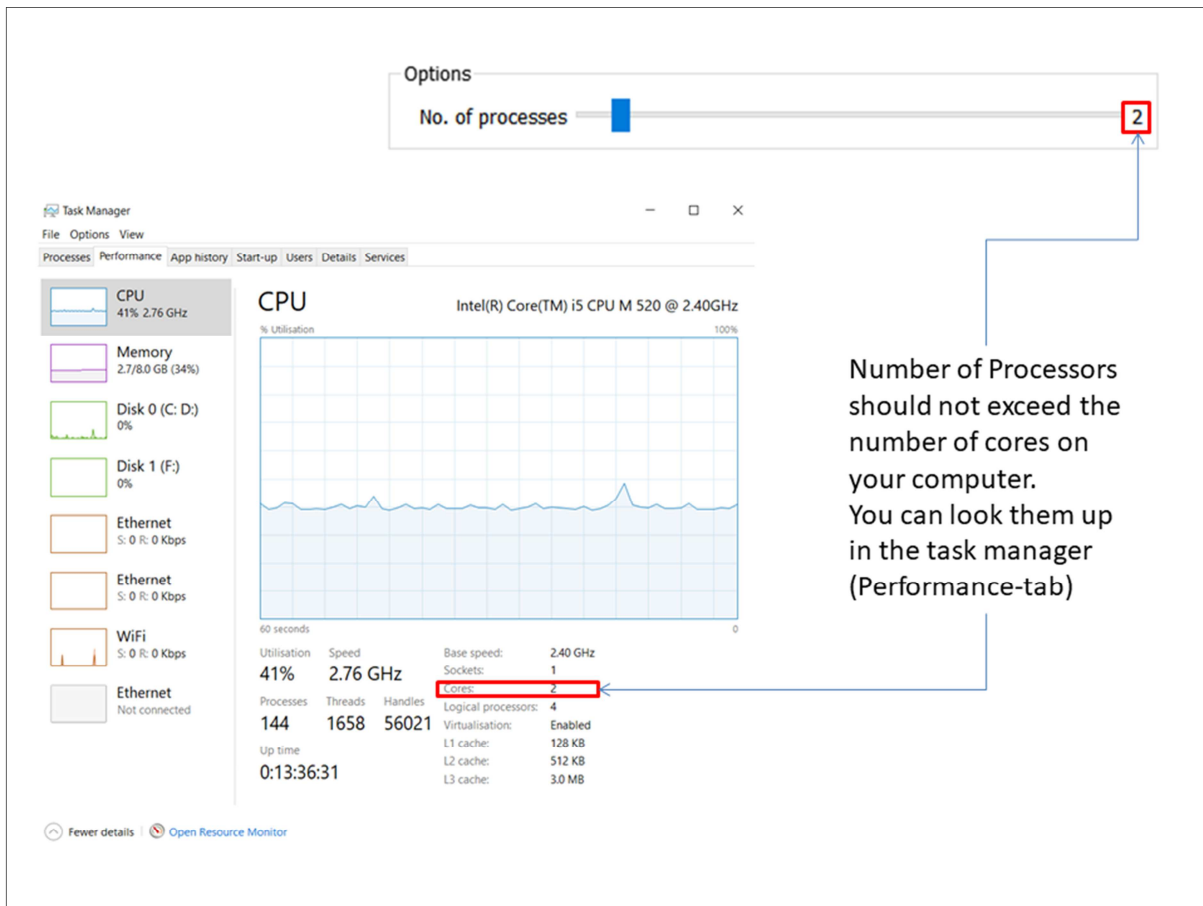
**Hardware: The most frequent problem** is insufficient hardware: If computations take an extremely long time (i.e. more than half a day per wind direction) or don't even seem to start, it is likely that your computer has insufficient CPU speed and/or RAM resources. If you have a standard office laptop it is even more likely to be insufficiently specified for OFWindCFD calculations!

**Other ways to avoid computer overload:** When you start simulations with multiple processors please make sure that the number of cores on your computer matches or exceeds the number of processes in OFWindCFD. If you cannot avoid using the computer during the computational run you may even want to reserve one core just for this work as otherwise all cores have to wait for the one core performing your other jobs on the computer.

If the number of processes is too high this has two consequences:

- a) Overload of CPU – this makes it slower than necessary
- b) As a consequence of the high number of sub domains the working memory of your computer might not be sufficient, it starts swapping and takes forever to run the case.

Please make sure you don't overload your computer (see next picture).



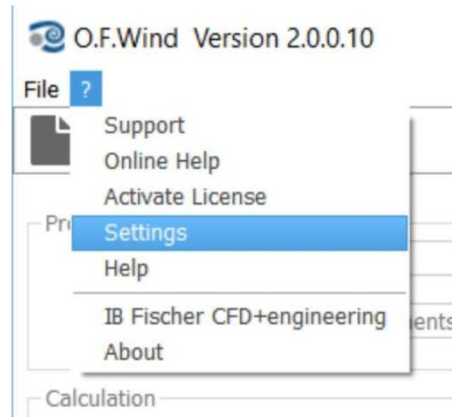
## 4.2 Temp directory has not enough disk space

The default directory where all data are stored during the simulations is the OFWindCFD working directory which is created in USER/AppData/Local/Temp/ on Windows 10.

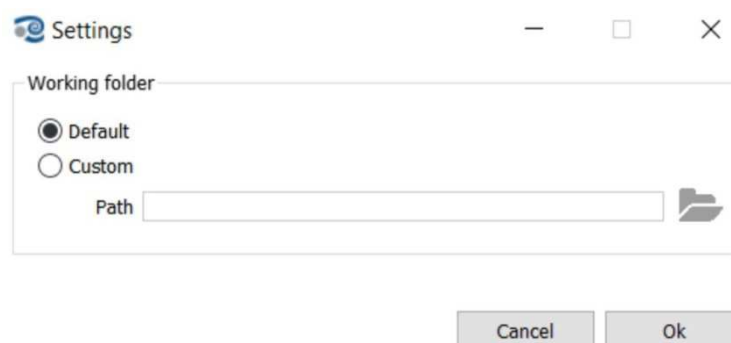
**This temp folder may run out of space and hence stop the computational run.**

**ATTENTION: In major Windows 10 updates the User/.../Temp directory is cleaned! So, if you save important data in this folder it will be lost. Please backup it somewhere or choose a working directory different from USER/.../Temp**

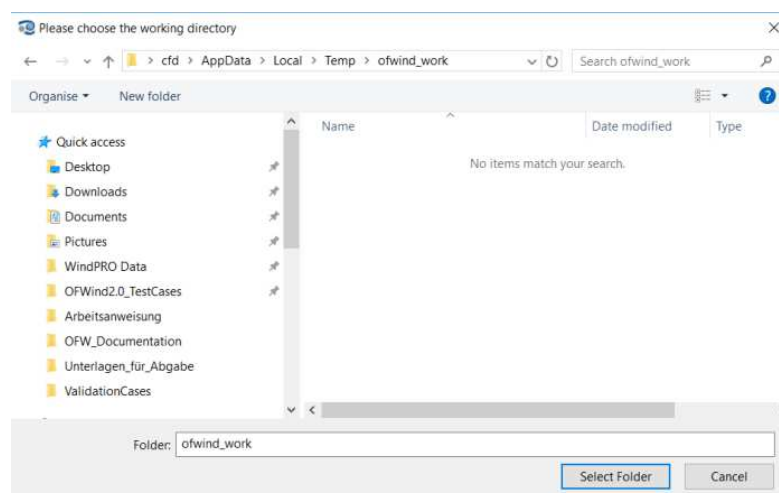
You can change the temp directory to a different place with more disk space available. To do this choose the “?” icon at the top and click on “Settings”.



In the next window change the selection to “Custom” and click on the folder icon. Then you can navigate to the directory of your choice.



Below is a screenshot shown with the default path of the directory “ofwind\_work”. The user name is “cfd”.





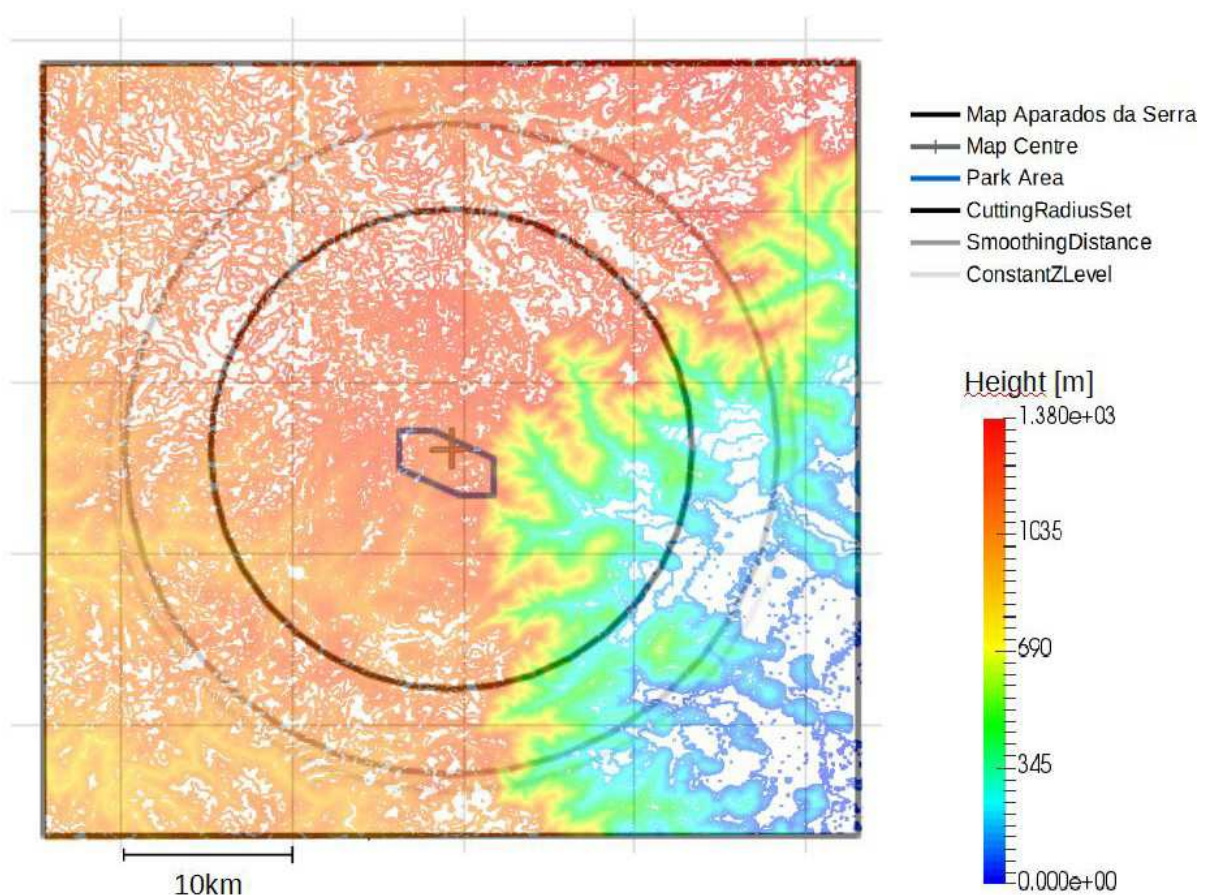
## 5 Technical Background – What is OFWindCFD?

OFWindCFD is a Tool to compute the flow field over a complex terrain. It uses the OpenFOAM® libraries and applications of version 2.3.x. Please note that OFWindCFD is not approved or endorsed by OpenCFD Limited, the producer of the Open- FOAM® software and owner of the OpenFOAM® and OpenCFD® trademarks.

OFWindCFD needs a flowrequest-Archive coming from windPRO with the following content:

- Orography map of your terrain
- Roughness map of your terrain
- xml-Document containing the coordinates of your WTG-Area and desired wind directions to compute (directions can be still altered within OFWindCFD)

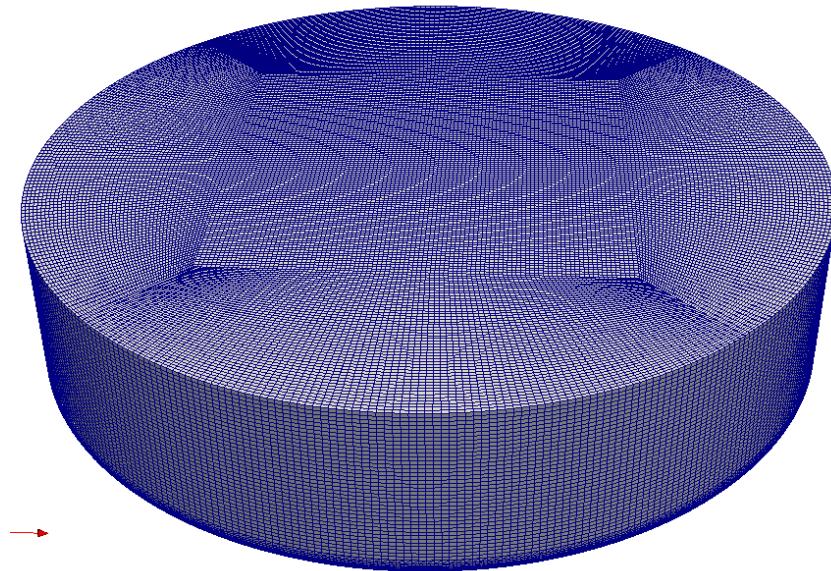
Based on the WTG-Area OFWindCFD will cut a circular patch from the terrain with the WTG-Area in the middle



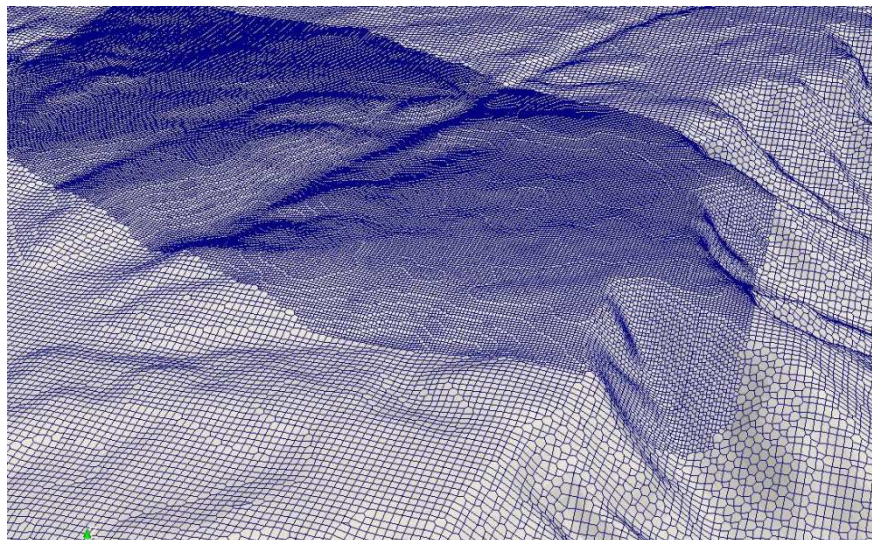
The user can select how fine the orography is resolved inside the park area (=WTG-Area) by setting the slider position to “Fast” (= coarser resolution) or “Accurate” (finer



resolution). Then OFWindCFD generates a fully-automated mesh which is needed to compute the flow over the terrain. The volume of the mesh resembles a cylinder to be able to compute the flow from each direction with the flow always perpendicular at the inlet. The terrain around the WTG-Area influences the flow inside the park area. This is the reason why not only the park area is used to create the mesh volume but also the surroundings within a certain radius.

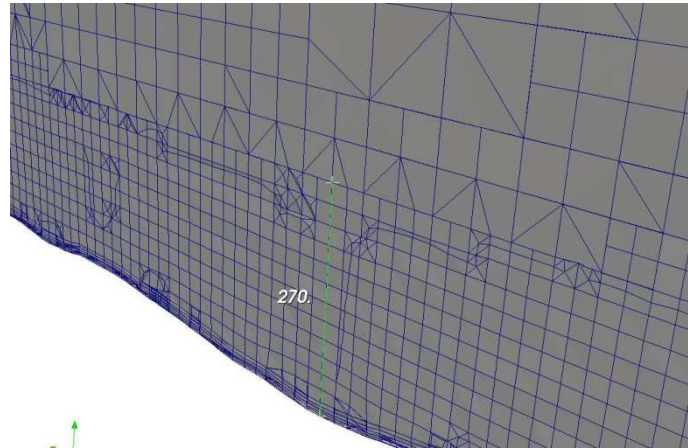


Inside the WTG-Area a higher terrain resolution is used to compute the flow here more accurately, and the grid gets coarser towards the outside.



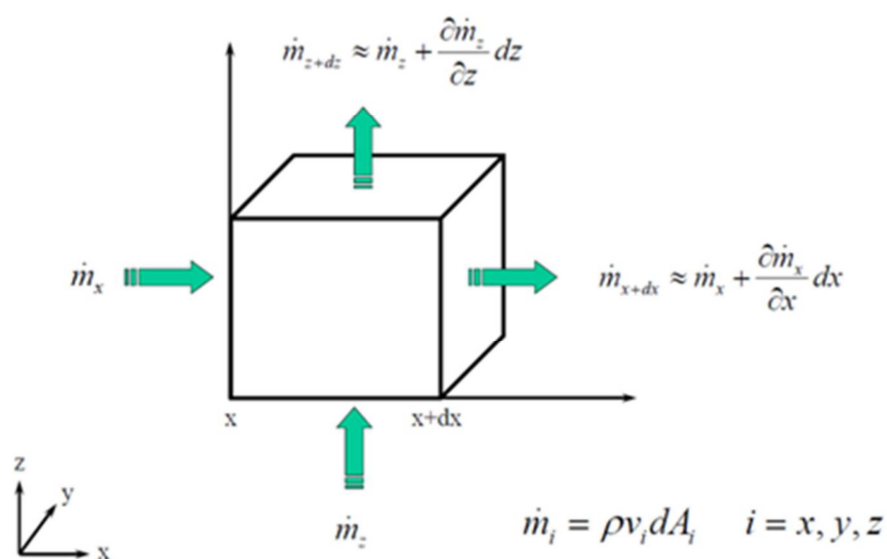
The next picture shows a vertical section through the grid in which you can see the

more detailed resolution in the first 270m above the ground.



By choosing the directions inside windPRO before exporting the flowreq-Archive or changing the choice in the OFWindCFD graphical user interface the user can decide which directions OFWindCFD is going to simulate. In the following paragraphs a short explanation summarizes how the flow over the terrain is computed inside OFWindCFD.

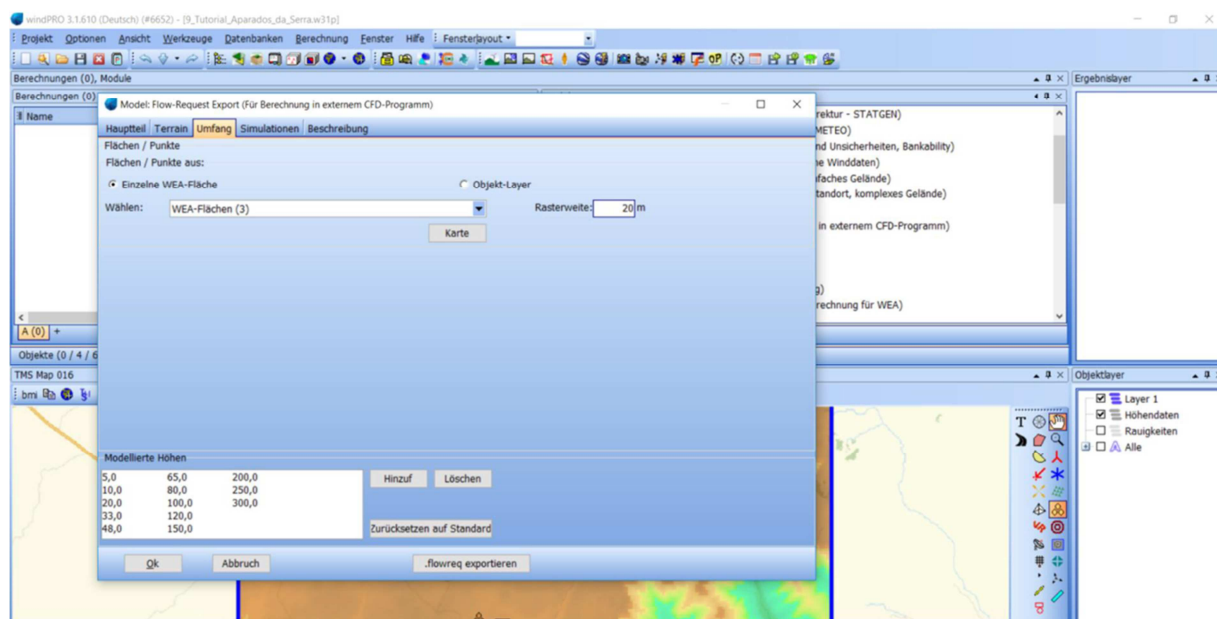
The method which is used in OFWindCFD to compute the flow over the terrain is based on the Navier-Stokes equations. The Navier-Stokes equations are a set of differential equations which can be used to compute the mass conservation, momentum conservation and energy conservation inside a given volume. Therefore, the volume has to be discretised into many separate volumes, the cells. Then the conservation equations can be solved in each of these cells. The picture below shows such a cell with the mass flow through four sides.



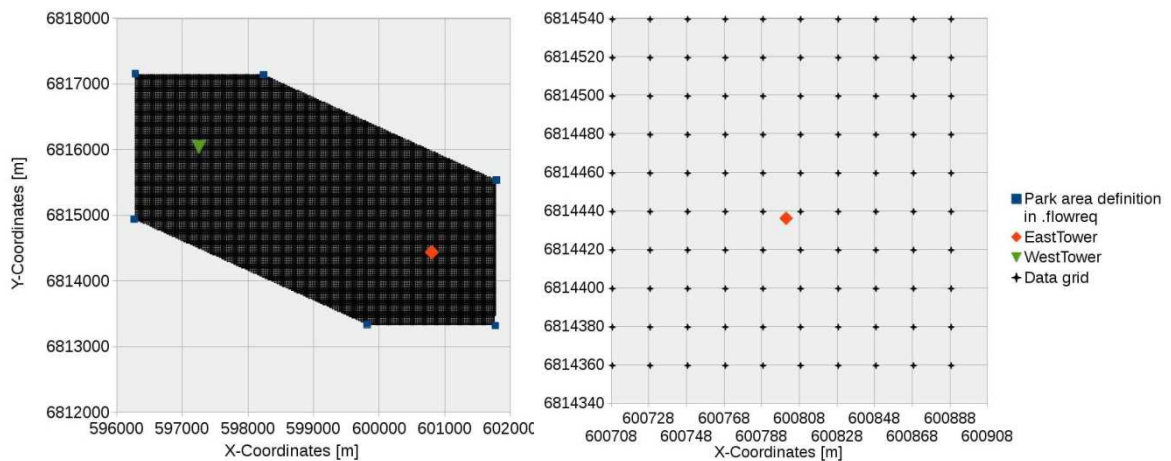
In order to be able to compute the flow field over the terrain on standard PCs in reasonable time, a couple of simplifications have to be assumed. These simplifications regard, for example, the turbulence modelling and the assumption that the flow field is in a steady state condition. For each cell, the mass and momentum conservation is computed throughout the whole domain. This is done in an iterative process so it takes longer the more cells are involved in the domain. This is why more accurate results always come with smaller resolution and therefore more simulation time. The iterative process of solving the equations in OFWindCFD for each cell ends when a steady state is reached.

As more explanation to the numerical simulation of flow would go far beyond the scope of this short introduction, the interested reader can look up these topics in literature like *Computational Methods for Fluid Dynamics* (1)

OFWindCFD computes all directions the user has selected in the scrollable pane. Before exporting the flowreq from windPRO the user has selected heights and a pixel size in windPRO (shown below) These parameters define a three dimensional grid above the WTG-Area which tells OFWindCFD where to sample the simulation data for the result files and should not be mistaken for the original CFD computation grid.



The picture below shows an example of the sample positions for a WTG-Area in 2D (left overall view and right zoomed into the east part).

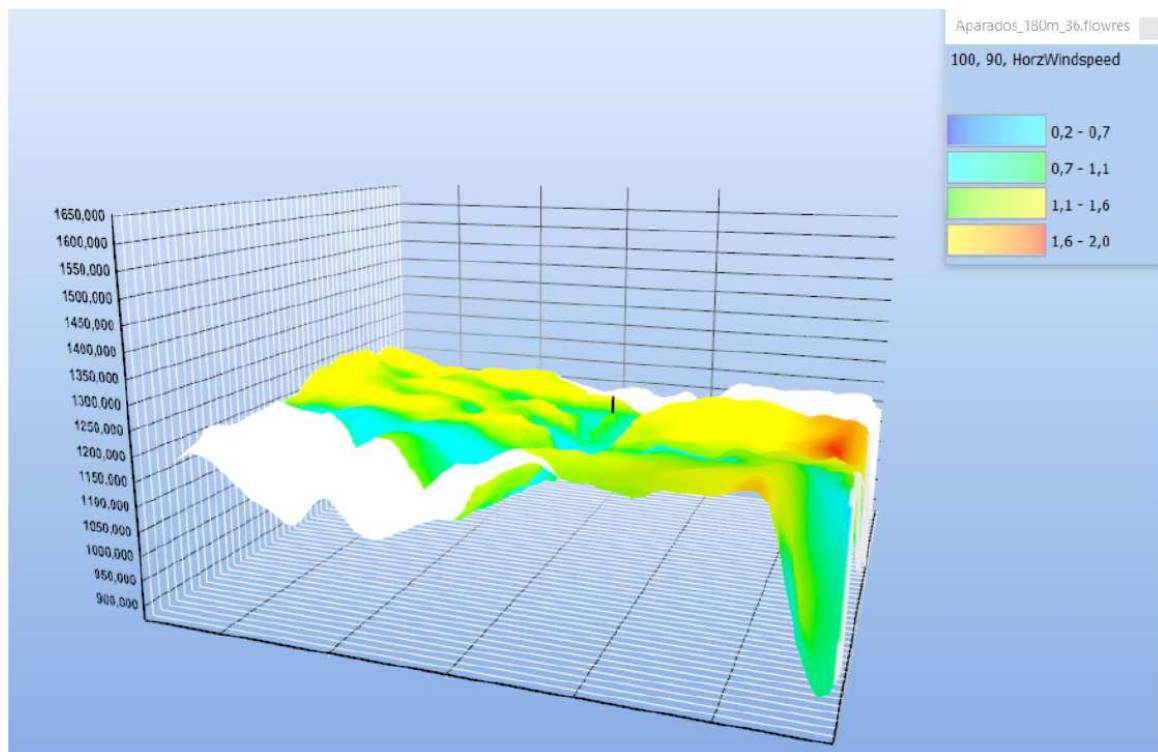


After each direction as been simulated, OFWindCFD stores the following data at each park area grid position into files called “SiteName\_Direction.flowdat”:

- Horizontal wind speed
- Turn
- Turbulence intensity (TI)
- Inclination

After 36 simulated directions (0°, 10°, 20° ... 350°) there are 36 flowdat-files with each having the information of the flow field above the WTG-Area. All flowdat-files are stored together into an archive inside the directory specified in the “Output” pane at the top of the graphical user interface. This archive is called “....flowres”. It can be imported back into windPRO to conduct predictions and analyse the flow field in the WTG-Area by the user.





## 6 References

1. **Joel H. Ferziger, Milovan Peric.** *Computational Methods for Fluid Dynamics*. s.l. : Springer, 2002. 3rd Edition.