Est USA 1981

Computational Fluid Dynamic Module

User Manual – November 2022

Introduction

AKCP's Computational Fluid Dynamics (CFD) module allows you to compute the air movement and its temperature all around the room using the probes connected. It integrates with AKCess Pro Server and is a natural extension that allows users to see what happens beyond the sensors. The document is split in five sections, from the most needed and useful information to the more obscure and specialized information. Following this logic the first section is a quick guide through the base, the second is a deeper description of the drawing process followed by instruction to install the CFD Server on your machines. For the advanced user, an additional section describes the modeling process and the physics behind it. At the end, there is an appendix with a troubleshooting section that might help you in case of difficulties.

We hope you will enjoy the CFD module!

Quick Reference Guide

Starting the Module From Nothing

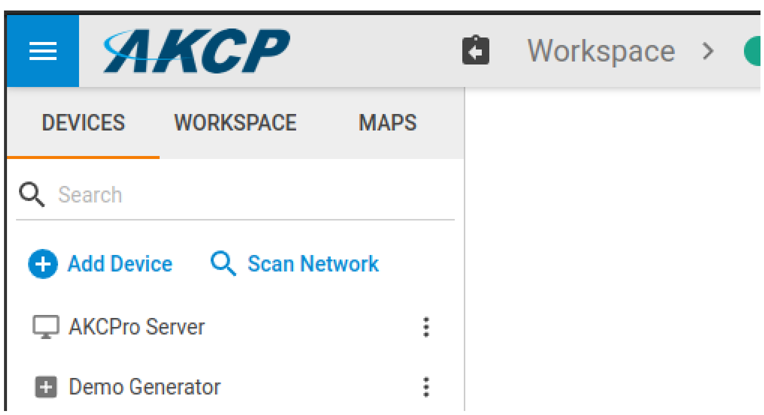
The module is designed to be used with the sensors provided from AKCP, but you can start using it from scratch. If some of the presented elements are already present on your server, feel free to skip to the next step. This section ex

Logging Into the Software

The CFD module is part of AKCess Pro Server (APS) installation. It can be installed on your local computer or on a server. In both cases, the service is accessible by a web browser, we support Firefox and Chrome. If the installation was made on a Windows OS, there is a link to directly access the right URL. In any other case, you can open your web browser and access the site by using the provided URL. If the installation is local, the URL will be 127.0.0.1:8080. For more details, please refer to the APS manual.

The Workspace and Maps

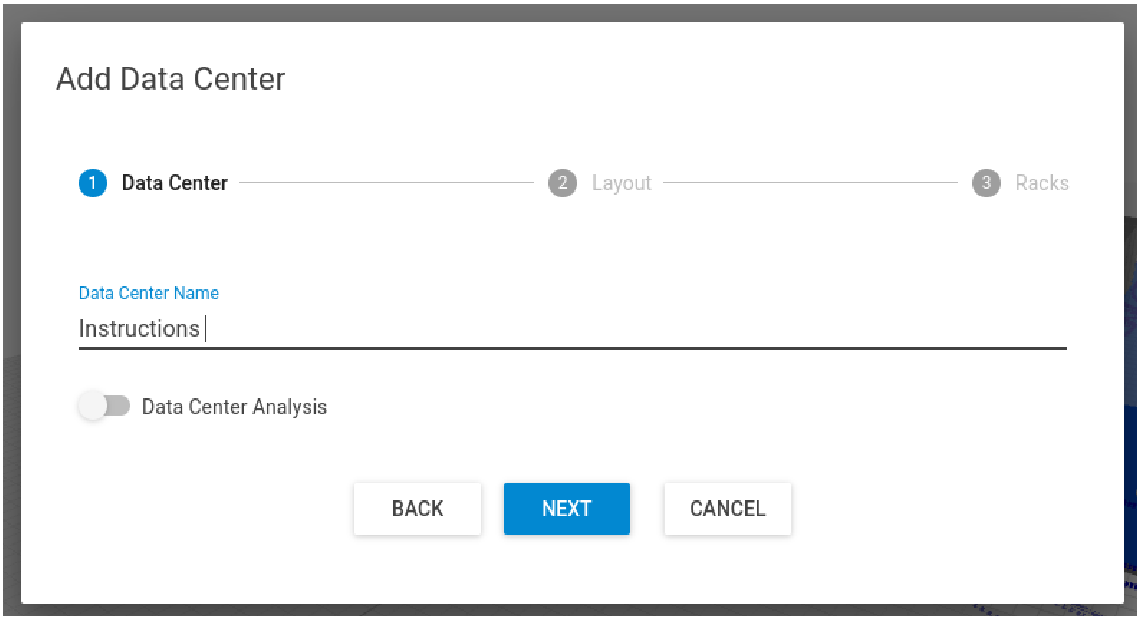
Once logged in, you will start with the main workspace. From there, click on *"MAPS"* on the top left of your screen. If the data center you wish to model is already present, you can skip to Section 4 *"Inputting the Value and Launching the Model"*. If it's not the case, you can click on *"Add Data Center"* to add a new one.



Adding a Data Center

Step 1: Data Center Name and Analysis Package

You are welcomed by a wizard that looks like the picture below. Choose your Data Center Name then click "NEXT". The name can be changed later.



Step 2: Data Center Layout

On this window you can choose a layout for your data center. You can start with an empty canvas or use a pre-built configuration, in a horizontal or vertical configuration.

In prebuilt cases, the aisle of RACK will be generated, with the containment and the doors. The RACK will be placed in the chosen axis and automatically paired. You can set the number of rows and columns. Then you can input the room width and length, and the gaps between the columns and rows if you use a preset grid. The “Unit” field allows you to select which units you are using for your measurements for this wizard.

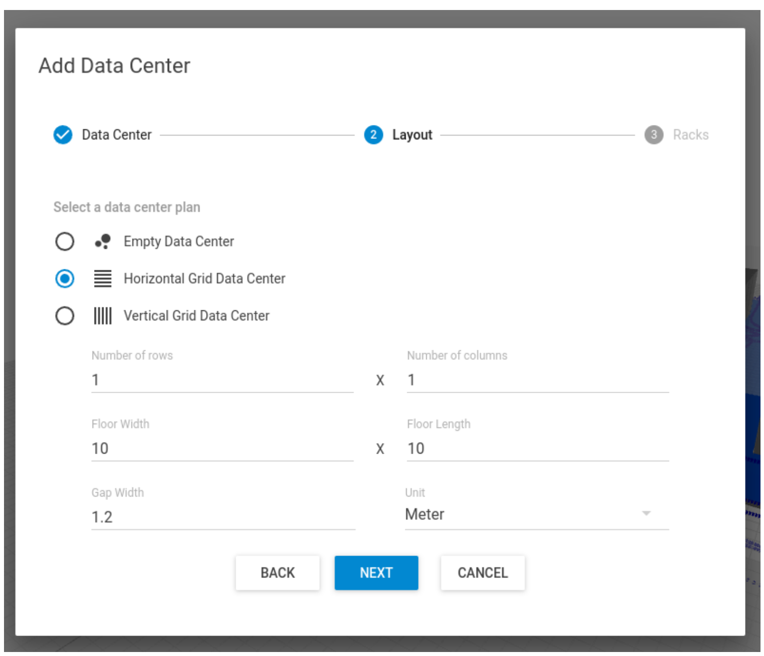
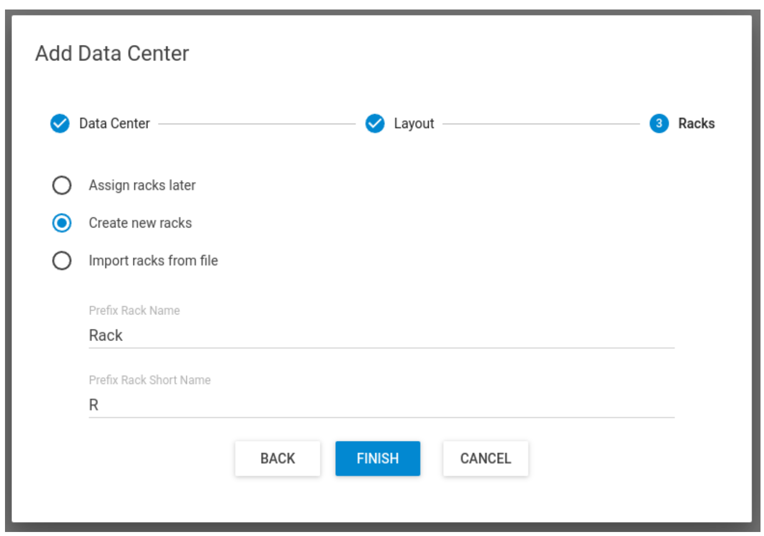


Figure: Vertical Grid Configuration (4 Rows x 4 Columns)

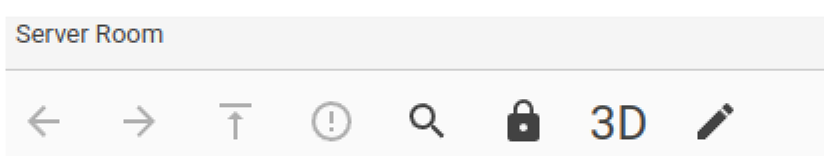
Step 3: RACKs maps assignment

The last step allows you to choose how to assign the maps of the RACK. The creation of maps for the RACK is covered in another help section. Here you can either choose to assign them later, create a new map for each rack generated automatically or import a CSV file with their configuration. If you just want to use the CFD module, you can choose *"Assign rack later"* without consequences.

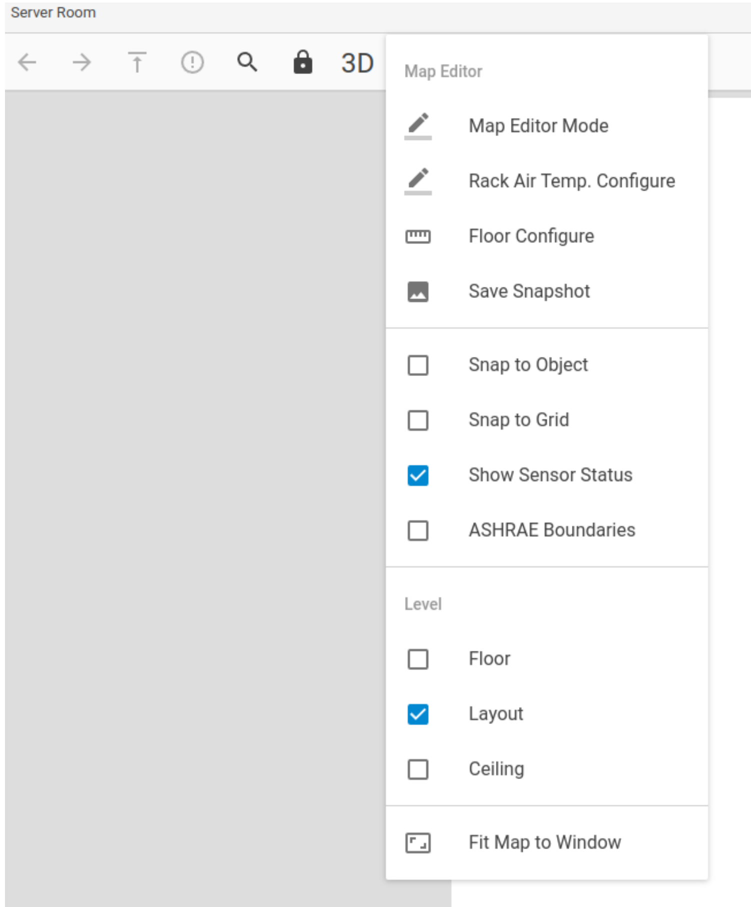


Drawing the Room

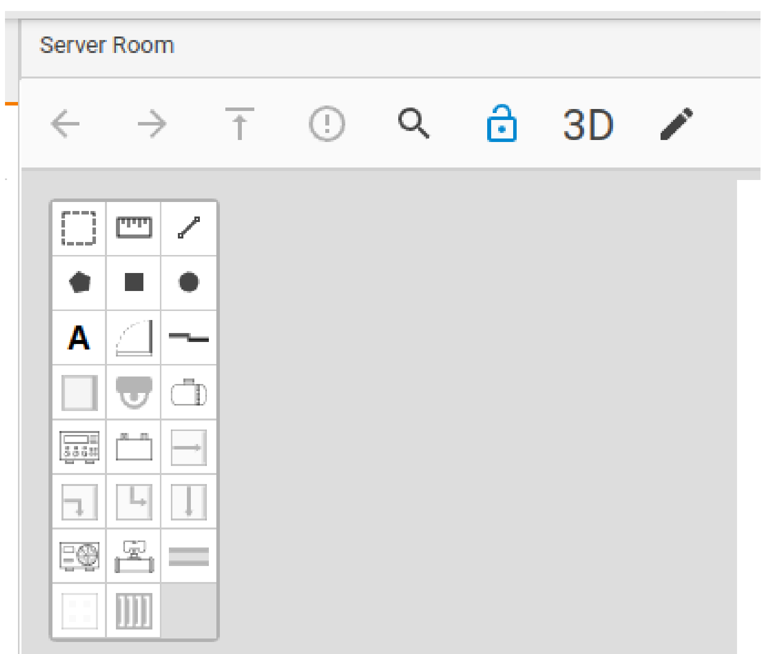
The drawing tool is made for a standard setup of RACK and CRAC. If your needs are different, please contact our support team. They will be happy to help you.



To start drawing, click on the lock to unlock the editor. Then click on the pen and select *"Map Editor Mode"*.

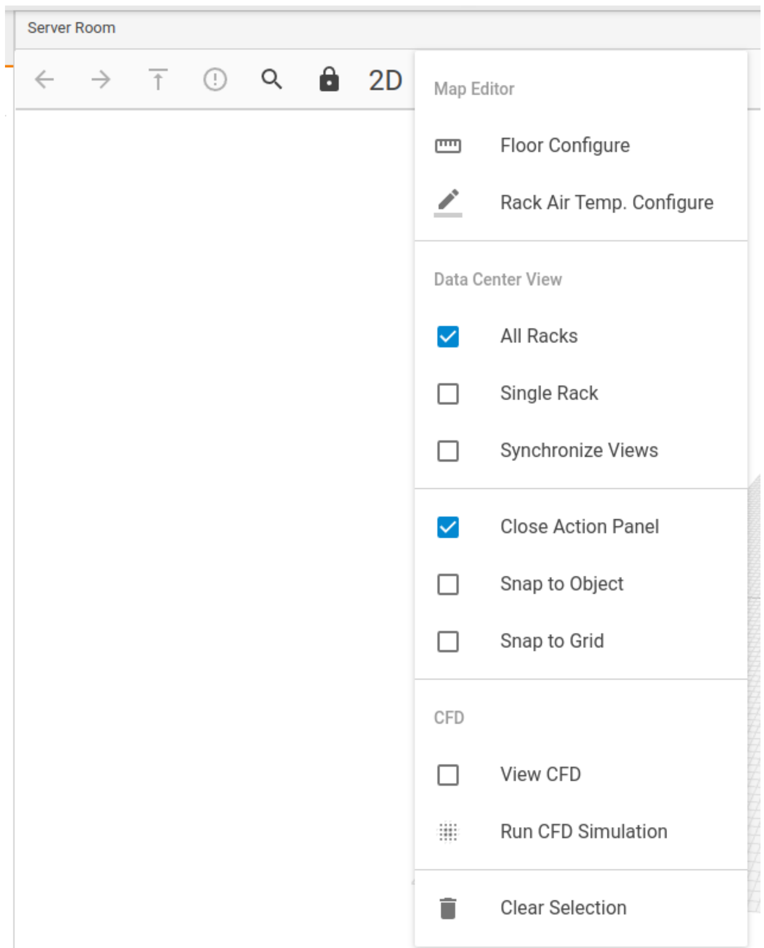


A toolbox containing all the available objects is displayed on the top left of the map. If the lock is blue and the menu is displayed, you are ready to draw.



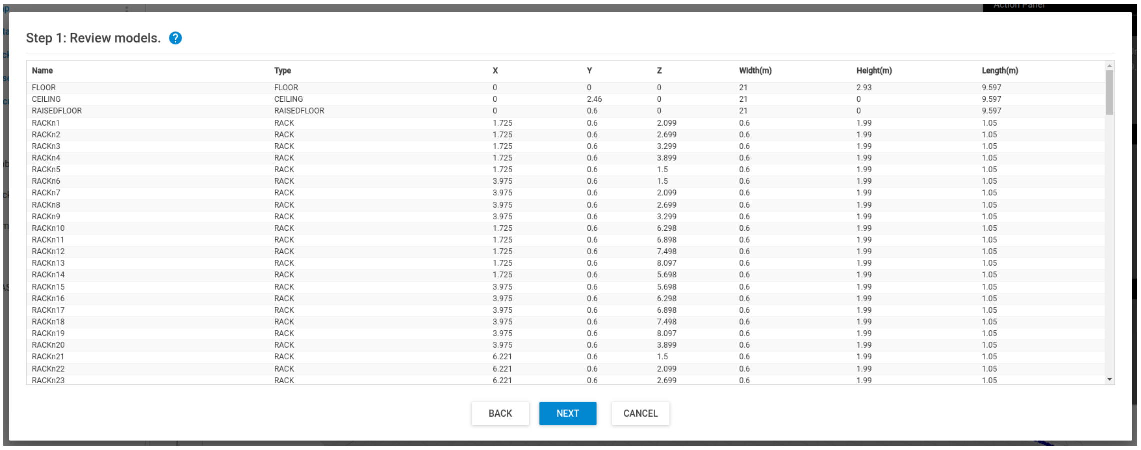
Inputting the Value and Launching the Model

When the map is ready, the next step is to launch the CFD model. To launch the CFD model, you have to be in the 3D view. Be careful, it's possible to edit the map in 3D, so be sure that the lock icon is closed and black. Once in 3D view, click on the pen icon, then select *"Run CFD Simulation"*. A wizard should appear.



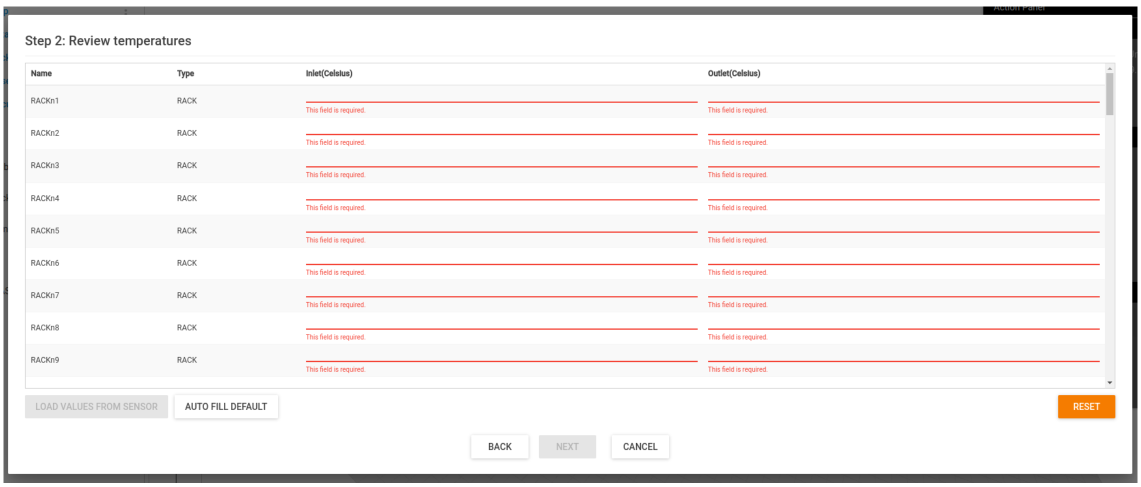
Step 1: Review Models

In the first step, you are presented with the list of the objects present on the map that will be used for the CFD model. Their name, type, location and size are all available in this list. You have nothing to change in this window, if everything is OK, you can click on *"NEXT"*.



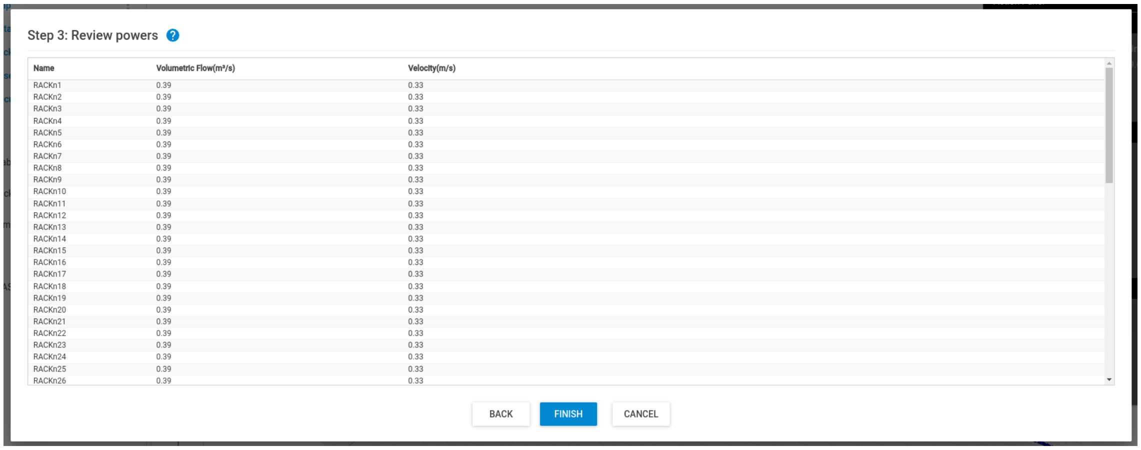
Step 2: Review Temperatures

In this step, you need to set the temperature value for each inlet and outlet of your RACK and CRAC. You can set each of them by entering the desired value in the corresponding fields. If a value is missing, the *"NEXT"* button will be grayed out. If you have probes for each inlet and outlet, you can use their current value by clicking on *"Load Values From Sensor"*. If the sensors are not available, the button will be grayed out.



Step 3: Review Powers

The last window displays the computed volumetric flow in cubic meters per second (m^3/s) and the velocity in meters per second (m/s). Those are computed by using the power and the temperature of the output. Carefully review the values, a badly drawn CRAC or RACK might have an abnormal number in the list. If everything seems fine, you can click on *"FINISH"*. Note that CFD computation will take a long time, please be patient.



Step 3: Review powers

Name	Volumetric Flow(m³/s)	Velocity(m/s)
RACKn1	0.39	0.33
RACKn2	0.39	0.33
RACKn3	0.39	0.33
RACKn4	0.39	0.33
RACKn5	0.39	0.33
RACKn6	0.39	0.33
RACKn7	0.39	0.33
RACKn8	0.39	0.33
RACKn9	0.39	0.33
RACKn10	0.39	0.33
RACKn11	0.39	0.33
RACKn12	0.39	0.33
RACKn13	0.39	0.33
RACKn14	0.39	0.33
RACKn15	0.39	0.33
RACKn16	0.39	0.33
RACKn17	0.39	0.33
RACKn18	0.39	0.33
RACKn19	0.39	0.33
RACKn20	0.39	0.33
RACKn21	0.39	0.33
RACKn22	0.39	0.33
RACKn23	0.39	0.33
RACKn24	0.39	0.33
RACKn25	0.39	0.33
RACKn26	0.39	0.33

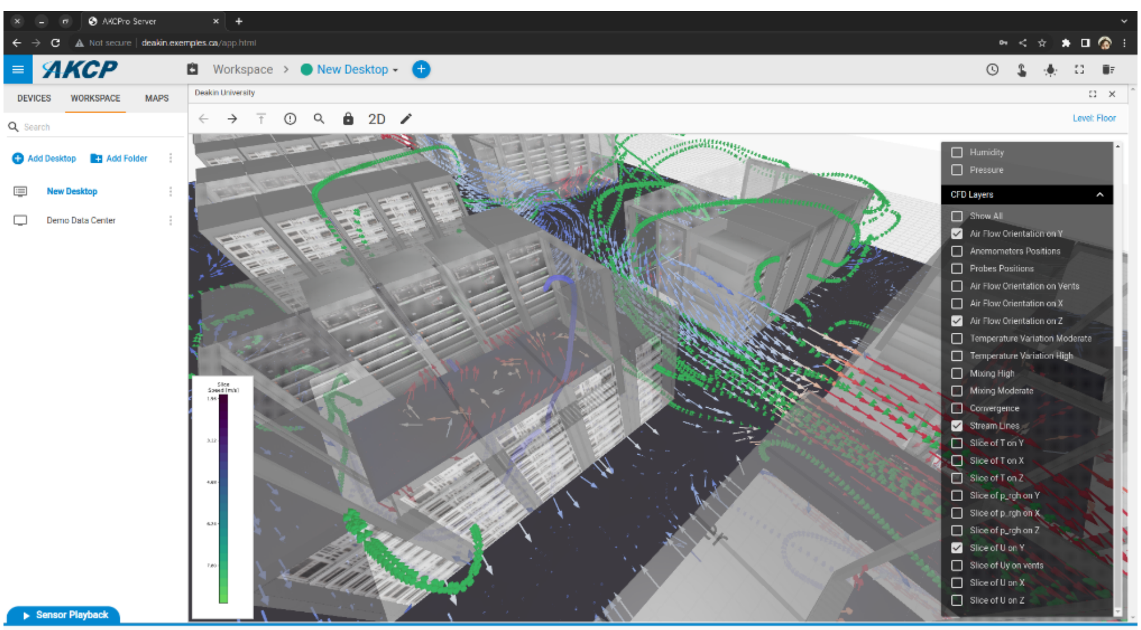
BACK FINISH CANCEL

Step 4: Confirmation

The last step is to confirm that you want to start the simulation. The server allows only one simulation at the time.

Looking at the Results

When the simulation ends, the results of the model can be seen on the 3D view. If they are not visible, you can click on the pen and tick the option "*View CFD*". Layers with different fields and display methods are available in the menu at the right of the screen. Since the menu is rather packed, the speed, temperature and pressure are shortened to their common variables: *"U"* (speed), *"T"* (temperature) and *"p"* (Pressure).



Layer	Description
Airflow Orientation on Y, X, Z or Vents	Arrow that shows the direction of the airflow.
Temperature Variation Moderate or High	Show zone where the variation of the temperature is high and probably abnormal.
Mixing High Moderate	Show zone where there is a lot of mixing.
Convergence	Red and Green sphere around the strategic location of the map. A green sphere means that the model seems to converge towards a solution and a red sphere means that the solution seems to have difficulties or is still not found.
Stream Lines	Line that follows the air displacement, like a weightless leaf floating in it.
Slice of T on Y, X or Z	Temperature map on the axes.
Slice of p_rgh on Y, X or Z	Pressure, including hydrostatic pressure.
Slice of U on Y, X or Z	Magnitude of the airflow speed map on the axes.
Slice of Uy on vents	Airflow Speed at the vents on the Up/Down direction. Positive being Up and Negative being Down.

More Information About the Drawing Process

This section has more description about the tool and how it's possible to use them. This tool is used in many applications, so, not all the items that can be drawn are included in the model.

Regarding the Drawing Process

To start drawing, the map must be unlocked by clicking on the black lock icon. If the icon is blue and unlocked, you can modify the map. The drawing mode can be activated by opening the menu and selecting “”. The toolbox should appear on the left of your screen.

To add an item, simply select it from the toolbox and draw on the map. Each item has their particularity, but they are usually drawn by clicking and sliding for their dimensions.

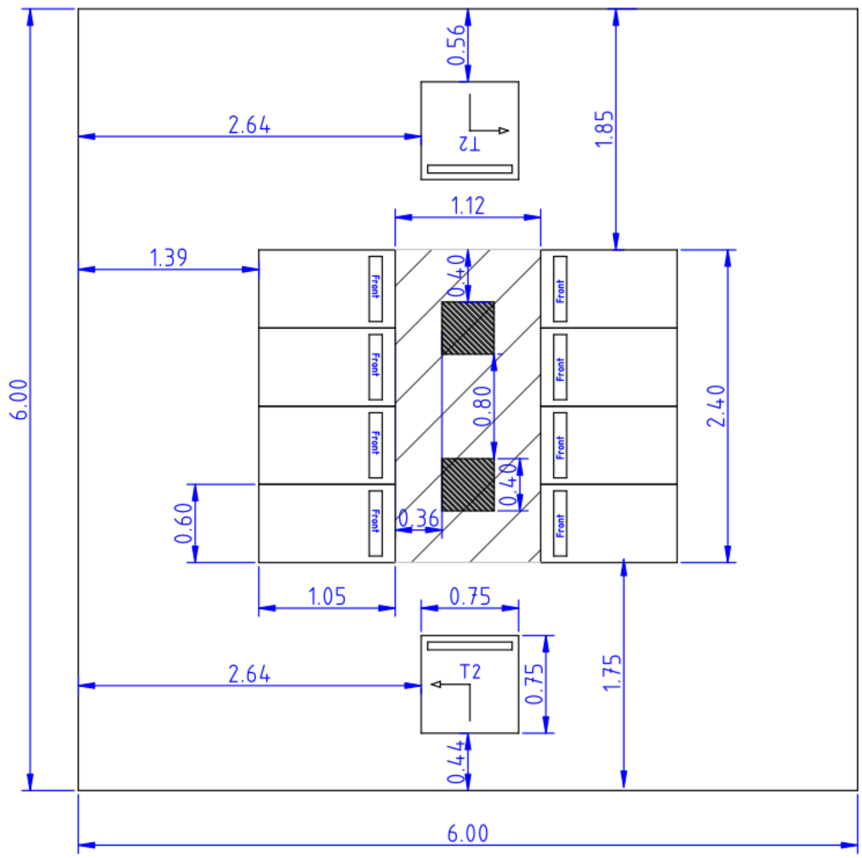
Usually the Cooling Units (CRAC) and RACK are the base items with the containment. The models don't need any containment, but they are quite common. The other items that can be included in the model are vents, doors and boxes.

The map is separated in three layers. The *“Layout”* level is the main level where most of the components go. The two other levels are *“Floor”* and *“Ceiling”*. In those levels, the user can place the vents. It’s possible to change the vents level using the right click contextual menu.

The toolbox in the top left contains all the items available for the level. Not all the objects will be included in the model. The recommended steps are the following:

1. Confirming the size of the Room. To change it, click on the *“pen”* button and click on *“floor”*.
2. Setting the default size of the elements in the room, like the CRAC and RACK height.
3. Adding the RACK to the “Layout” layers.
4. Adding the vent on the ceiling and floor layers.
5. Adding the containment box.
6. Adding the sliding doors.

Basic Example: Cold Containment



The following is a step-by-step tutorial to draw a single aisle of a cold containment unit. The goal is to reproduce the setup shown in the figure presented above. This map was used and built in a presentation available online: https://youtu.be/2TGRqY4p6bU?t=854

1. Create a Datacenter map. Since it will be a cold contained map, you can name it *"Cold Containment Demo"*, select *"Empty Data Center"*, then click on next.

2. In the *"Layout"* form, enter the room dimension as follow:

   a. Room Width: 6 m
   b. Room Length: 6 m
   c. Room Height: 2.6 m
   d. Ceiling Height: 2.2 m
   e. Raised Floor Height: 0.4 m
   f. Rack Height: 1.7 m

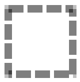
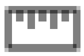
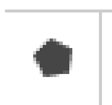
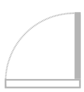
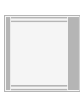
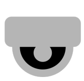
3. Unlock the drawing by clicking on the *lock icon*. If the lock is blue and open, the map is unlocked.

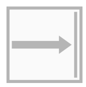
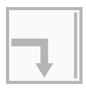
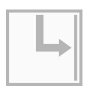
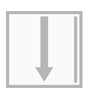
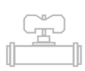
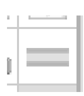
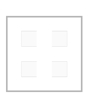
4. Activate the *"Map Edition Mode"* by clicking on it in the *"pen"* menu. If a toolbox is available on the left of your screen, you are in *"Map Edition Mode"*.

5. Select *"Rack"* in the toolbox at the left of the screen.

6. Place the first rack in position. Use the arrow on the keyboard to make small modifications.
7. You can adjust its dimension by right-clicking then selecting *"Edit"* in the contextual menu. The rack will be 1.05 m by 0.6 m.
8. Activate the *"Align on Object"* option, by selecting it in the *"pen"* menu.
9. Duplicate the rack and align it with the first one. To duplicate, open the *contextual menu* by right-clicking the object, then select *"Make a Copy"*. Repeat the process until a column of 4 racks is created.
10. Select the *containment tool* in the *toolbox* at the left of the screen.
11. Slide from the corner of the rack to approximate the dimension.
12. Right click on the containment and change the dimension to fit perfectly.
13. Copy one of the racks.
14. Open the *contextual menu* by right-clicking on the object and in the menu *"Flip"* select *"180°"* to flip the rack orientation.
15. Align the rack with the containment.
16. Duplicate the rack and fill the 3 other spots, filling the right side of the containment.
17. Select the *CRAC tool* in the left *toolbox* labeled *"Back In Bottom Out"*.
18. Place it close to the desired position using the plan.
19. Right click to open the *contextual menu* and adjust the dimension of the CRAC. In this mode, the CRAC is 0.75 m by 0.75 m.
20. Right click on the CRAC to open the *contextual menu* and in *"Rotate"* select *"90°"*.
21. Place the CRAC to its exact location, use the arrow on the keyboard to make small modifications.
22. Copy the CRAC and place it in the second location
23. Right click on the CRAC to open the *contextual menu* and in *"Flip"* select *"180°"*.
24. Select *"Containment Vent"* in the *toolbox* at the left.
25. Place the vent and adjust its size using the *"Edit"* option of the *contextual menu*.
26. Place the second vent by duplicating the first one. To do so, right-click on the object and, then, select *"Make a Copy"*.
27. Move both vents by right-clicking on the, then in *"Move"* select *"Floor"*

The Toolbox Reference

Icon	Name	Description	In Model
	Select	Selection tool. Click and drag to select multiple elements.	No
	Ruler	Measurement tool. Displays the distance between the points. The ruler doesn't alter anything on the map.	No
	Line	Draws a line. Clicking will add a new point on the line, to end it, close the polygon or click on "Finish". The setting of the line can be changed by clicking on it once drawn.	No
	Polygon, Square, Circle	Draws a shape. The setting of the geometry can be changed by clicking on it once drawn.	No
	Static Text	Write a label on the map. You can freely place the text zone by clicking and dragging. Double click on the label to edit the text.	No
	Door	Draws a door.	No
	Slide Door	Represents a sliding door. This element will be included in the model. Its default height matches the default RACK height.	Yes
	RACK	Represents a RACK. The map of a RACK (RackMap) can be assigned to this item. The RACK must have a temperature and power value for the modeling.	Yes
	Camera	Represents a Camera.	No
	Fuel	Represents a Fuel Tank	No
	Generator	Represents a Generator	No

	Battery	Represents a Battery	No
	CRAC Type 1	A cooling unit with a back to front configuration.	Yes
	CRAC Type 2	A cooling unit with a back to underfloor configuration. The floor will be cut to open the outlet of the CRAC. The temperature and power of the CRAC must be defined to allow the model to run.	Yes
	CRAC Type 3	A cooling unit with a ceiling to front configuration. The ceiling will be cut to open the inlet of the CRAC. The temperature and power of the CRAC must be defined to allow the model to run.	Yes
	CRAC Type 4	A cooling unit with a ceiling to underfloor configuration. Both the ceiling and the floor will be cut to open the inlet and outlet of the CRAC. The temperature and power of the CRAC must be defined to allow the model to run.	Yes
	Air Conditioner	Any other kind of cooling unit type.	No
	Valve	Represents a valve.	No
	Pipe	Allows you to draw piping.	No
	Containment	A space contained between walls. The vents for RACK and CRAC will be cut out from it.	Yes
	Vents	A vent. The vents can be placed on the floor layer, the layout level and the ceiling. **Important:** The vents placed on the LAYOUT LEVEL are called "Containment Vents" and will automatically cut and create the duct to the ceiling.	Yes

Installing the CFD Server on your machine

A Docker image of the server is available. The CFD server can be installed on the same computer as APS. But, keep in mind that CFD modeling requires a lot of resources. Therefore it is recommended to install the server on a dedicated remote machine.

Advice: The best practice for running a few models with ease is to start a machine with Docker preinstalled on a cloud computing service. Most of those services are billed for the time used. You can rent a really powerful machine for less than a dollar per hour and the configuration is easy.

Starting the Server on Linux Console

On most Linux flavors Docker is included in their package manager. To install the software on Debian/Ubuntu you can use the following command line:

```
$ sudo apt install docker.io
```

You can also download the software from docker website: https://www.docker.com/

The server image contains all the required software and will start the server on port 8089. To start the service, use the following command line:

```
$ docker run --rm -u CFDUser -e PORT=8089 -p 8089:8089 cfdserver/cfd-server
```

Docker will download the image on the first use, it's a full server so it can be long, please be patient. You can change the port number and the access token values. Please, be sure that the ports you desire to use are allowed on your firewall. If you change the port number, be sure to update the published port of the docker accordingly. “--rm” will remove the cont

ainer when closed.

You can create a script or change the option so that the server restarts automatically with the hosting computer. For more information about docker run command: https://docs.docker.com/engine/reference/run/

How to Stop and Manage the Container on Linux

The server is supposed to stop when quitting the task on the terminal (CTRL+C). If the window is closed before exiting, the container will keep running. It can be closed by using the docker command lines interface.

To see the active container:

```
$ docker container ls
```

To stop the cfdserver/cfd-server container:

```
$ docker container stop [containerId]
```

Starting the Server on Microsoft Windows

If you use Microsoft's Windows or Apple's Mac OS, you might prefer using the desktop version of docker. Please, keep in mind that the modeling process is really demanding on resources and that it's usually run on a dedicated computing machine. It's really important that you use a different computer than the one hosting AKCPro Server. Also, please note that the Microsoft's Windows installation uses extra compatibility layers, which make the service consume extra memory.

1. Download Docker Desktop for Windows from the website: https://www.docker.com/
2. At installation, select *"Use WSL2 instead of Hyper-V"*
3. Reboot the PC when asked
4. Install the WSL2 MSI package: https://learn.microsoft.com/en-us/windows/wsl/install-manual#step-4---download-the-linux-kernel-update-package
5. Install a Linux distro package for WSL2. We recommend Ubuntu 22 LTS: https://aka.ms/wslubuntu2204
6. Restart the Docker service to recognize WSL2 and the installed Linux (no need to reboot)
7. Open Docker Desktop
8. Open Settings / Resources and enable "Integration with the Ubuntu distro" option
9. Open PowerShell and run command:

```
$ docker run --rm -u CFDUser -e PORT=8089 -p 8089:8089
cfdserver/cfd-server
```

   It will download the files from Docker repository and run the CFD server. This is a server image of about 30 GB, therefore the download can take a while. Please be patient.

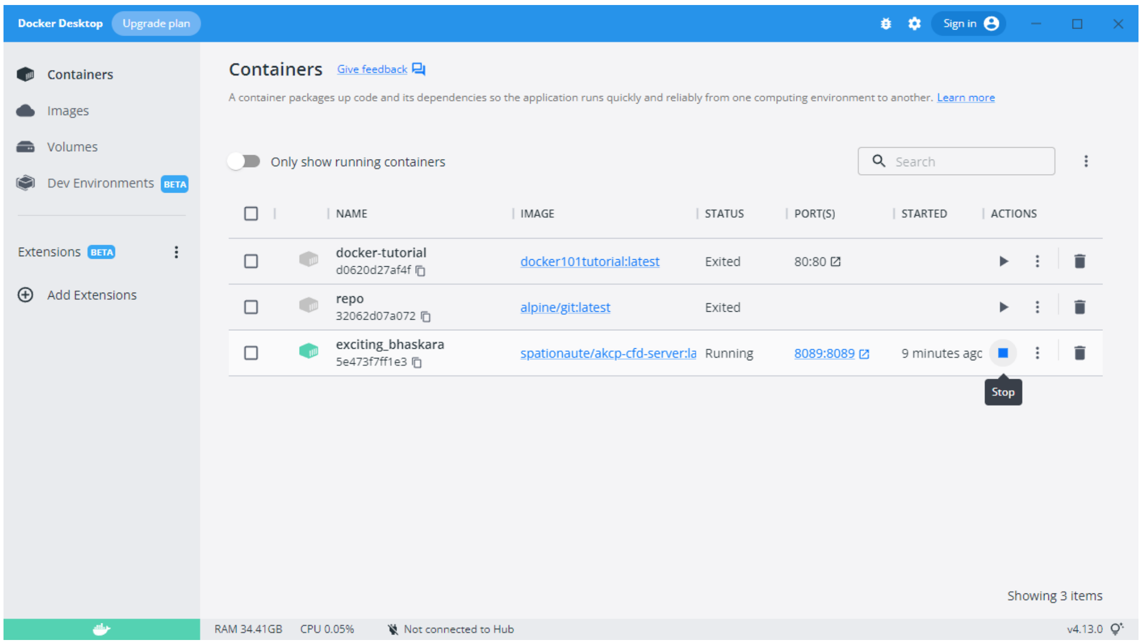
10. After the image is downloaded and started, you can monitor it with Docker Desktop under *"Images"* and *"Containers"*
11. The started CFD server can be stopped from Docker Desktop GUI: stop command under *"Containers"* or close the PowerShell window
12. To update CFD server image, choose the command *"Pull"* under the *"Images"* from Docker Desktop GUI
13. CFD server can also be started from the GUI (from *"Images"* menu), but it will consume more disk space

Manual installation steps for older versions of WSL: https://learn.microsoft.com

Advice: WSL2 will continue to run until closed. If you don't need it don't forget to close WSL2 when finished.

How to Stop and Manage the Container on Windows

If the image fails to be closed from the PowerShell, it’s possible to use Docker Desktop to close the container. First, select *“Containers”* then look for the container with the image *“akcp/cfd-server”* then click on stop.

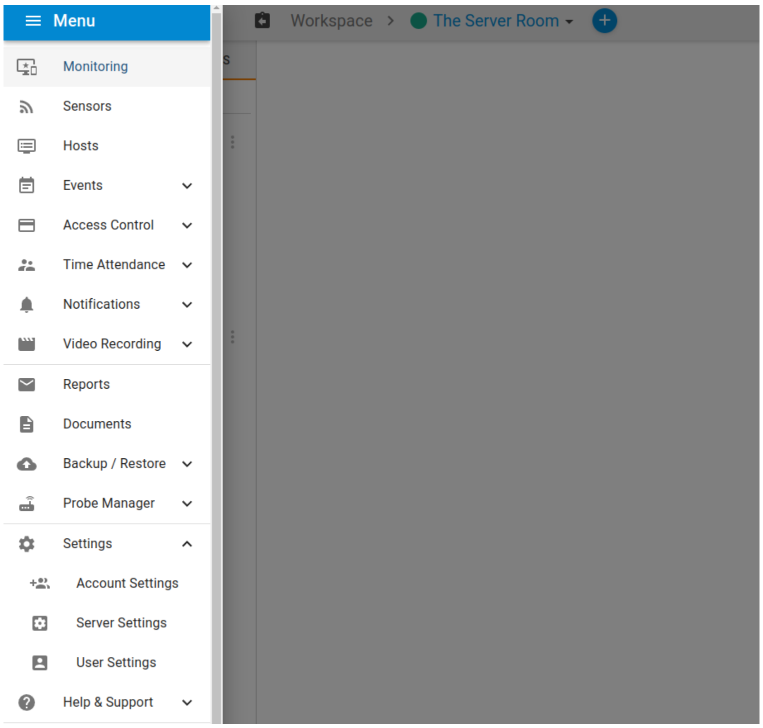


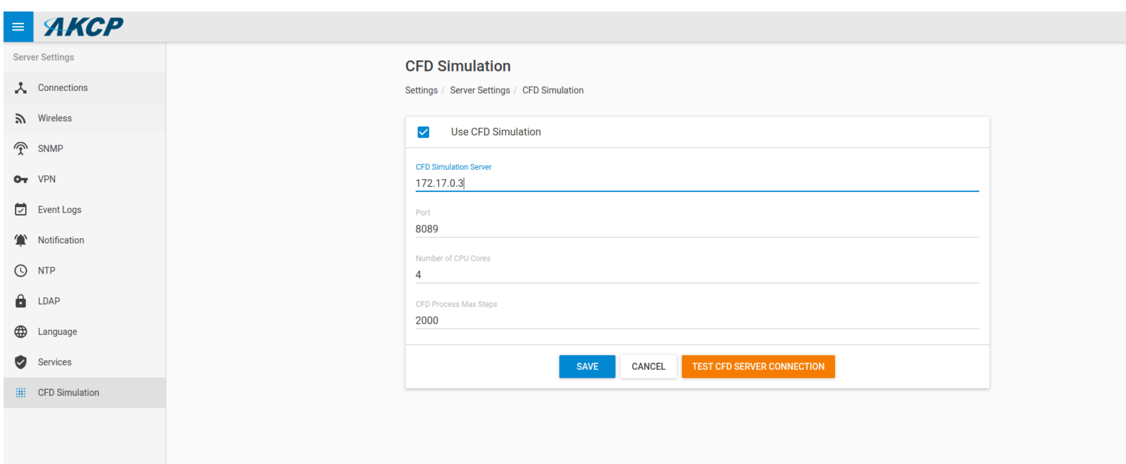
It is also possible to manage the image of the server by clicking on *“Images”*. In this menu, by clicking on the three dots, you can inspect, remove or update (pull) the server image.

Advice: When set up with WSL2, all files are stored inside a compressed VHDX image file under: `%UserProfile%\AppData\Local\Docker\wsl`

Activating the CFD Module on AKCPro Server

When the server is ready, the CFD module can be activated on the AKCPro Server software. The module is accessible by entering the *"MENU"* and choosing *"Server Settings"* in *"Settings"*.





Recommended Hardware For the Server

It is recommended to run the server on a second computer. Many companies offer high-performance computing machines that can be rented by the hours. The parallelization is one of the keys for speed, so a system with a high number of cores would perform the computation much quicker. While lower-end systems can offer the same accuracy at the cost of time.

The CFD Server is made to be deployed on computing platforms, it’s possible to deploy it on Windows but with an overhead cost in RAM.

Operation System	Linux
CPU	64 bits, with KVM (Virtualization)
Memory	16 GB of RAM (larger model might need more)
Image Size	~30 Gb

Operation System	Windows 10, Windows 11
CPU	64 bits, with KVM (Virtualization)
Memory	32 GB of RAM (larger model might need more)
Image Size	~30 Gb

More Informations About the Modeling Process

This section is a deeper description of the modeling process. Reading it is optional, but it can help you understand the reason behind some obscure bug or strange behavior that will happen sometimes when running the simulation.

The model computes the air velocity and its temperature using a well-documented toolkit that is used in many industries. There are 4 important steps that are automatically made by our software.

1. Surface and Objects Computer Drawings
2. Segmentation of the space (Meshing)
3. Iterative modeling
4. Post-Processing and Display

Surface and Objects CAD

In this first step, the software will prepare the RACK, the CRAC, the floor and the ceiling. If there are ducts from the containment to the ceiling, they will also be drawn. The surface will be used in the next step.

Segmentation of the Space

This is a really important step for the accuracy, speed and quality of the result. The 3D space is split in little sections that are easy to compute. Those sections have quality, we did our best so that quality is the highest in most situations. But sometimes, drawing can cause problems.

Iterative Modeling

The final step is the simulation itself. The program will enter in a loop of iterations in attempts to solve a large series of equations. When a loop is completed, it's called a step. The server is by default limited to 2000 steps and can exit automatically when all the variables are stable. Solving a model can take up to 4 hours on a good machine; therefore it's recommended to use a dedicated installation. You can use our Docker image on AWS, or follow the instructions in the section "Installing the service on your own machine."

Regarding the Model Physics

This section is a description of the physics used in the model. It's destined for advanced users or to answer some questions about how we reach the results.

Limits for the Model

The following table shows the limits for each input of the model. The UI will warn if one of the values is outside of the recommended range.

Variable	Units	Warning		Error	
		Min	Max	Min	Max
U	m/s	-30	30	-273	273
P	W	0	360000	0	150000000
T	°C	-40	75	-274	1200

Algorithm

The model uses the *Finite Volume Method* (FVM) which is considered the most precise approach available for fluid dynamics. The turbulence model uses the *Reynolds Averaged Stress* (RAS). Once launched, the program solves the continuity equation, the momentum equation and the heat transfer equation. It does so, following these steps:

1. Do a first approximation of the momentum
2. Solve the pressure with the approximation
3. Correct the momentum
4. Solve the heat transfer

The velocity for the RACK and the CRAC outlet/inlet are computed using the following equation:

$$\text{Volumetric Flow} = \frac{\text{Power}}{1278 \times \Delta T}$$

Constant Used for Air

Name	Value	Unit
Air Viscosity	1.5e-5	m^2/s
Air Density	1.204	kg/m^3
Air Heat Capacity	1006.38	J/(kg·K)

Troubleshooting

- **The simulation seems to crash at a step called “Meshing”.**
- **There is a warning that the meshing is bad.**
    - The meshing is the step where space is divided into small sections. If this step fails, try to change how you draw things. It’s possible that an object is too small or that something unusual is in the drawing. Look for gaps and miss aligned RACK. Also, look out for gaps between the RACK and the containment.
- **The model crash and the cfd server display “EXIT CODE: Null”.**
    - Those crashes are often related by variables that overflow. The software is supposed to catch those problems, but sometimes all the processes are shut down. To fix it, first look for any object, like CRAC or RACK that could be superposed or wrongfully oriented. Then, expand the containment a bit to be sure they overlap the RACK. It’s better that they overlap than fall short. Finally, if nothing is fixed, look for inlet/outlet proximity. If they are really close and their velocity high, they might generate some problems.
- **I have two really close objects and the model does not compute that region.**
- **The model seems to join the objects together.**
    - The model has a resolution limit that follows the importance of the region. When two objects are really close and the region is of lesser importance, they will be considered close. If this region is important for you, we can change the setting to make the software more precise at the cost of computing speed. Here few examples:
        i. Ceiling and RACK,Containment or CRAC1,2 at less than a decimeter.
        ii. Two RACK at less than a centimeter side by side.
    - It’s also possible that the gaps are computed, but not displayed. Look for mixing and velocity differences around the gap.
- **The Velocity Map is uniform.**
- **My arrows all have the same color.**
- **The highest velocity on the scale is really high.**
    - The boundary conditions are probably wrong or the simulation needs more steps to converge. If the velocity seems correct in the verification step, look for airflow going in the wrong direction (generally into an outlet). If you are representing a real setup and see an arrow pushing back in the outlet of a RACK or CRAC, there is probably a backflow. A backflow will lower the efficiency of your setup. You can use the “Large Temperature Variation” to help you find the problematic inlet.
        i. **Tech Note:** The velocity of an inlet/outlet is set to be a mean then computed on its surface. If an important backflow happens, the model will compensate at the extremity of the inlet/outlet with a really high outbound flow.
- **I have a strange empty box.**
    - Sometimes when drawing it’s possible to put objects that are really small and hard to see in the drawing. To spot them, you can use the list of objects that is presented in the first window of the CFD assistant.
- **The center of my containment has no results.**

- This happens when a section of the model is completely separated from the rest. Please look if the duct and the vents are connected to your containment. To create a duct, you need to use "CONTAINMENT VENTS" available in the layout layer.

- **There are small holes in the corner of the simulation:**
  - Those are normal and they are there to ensure that the computational domain is connected with the inside of the ceiling and the underfloor.
- **I get a strange gradient on my inlets:**
  - The inlet temperature is fixed to the value that you input or the value of the sensors. If a strange gradient exists, the model has difficulty to solve the reason why the temperature is lower/greater than the air reaching it. If this happens when you are using the sensor's value, it means that your model does not represent reality. In other words, something is not in the model! In that case, look for leaks in your containment or gaps that are not included in your drawings. **Finding those differences can save you money!**
    i. **Tech. Note:** Usually the inlet temperature values are not set since the heat is transported by air entering it. But our model is designed to work with sensors that, in all cases, represent reality. So we decided to fix the value of the sensor. This allows the user to detect discrepancies between the model and reality. If you use the sensors, look closely at the inlet's results, they can help you find a lot of problems.
- **I need help to install the CFDServer on my machines:**
  - Since the module is free, we expect you to do most of the setup work. But, if you need help, we will gladly help you for a reasonable fee.

FAQ

- **How do I update the server?**
  - To update the server you don't need to download everything again. Simply run:

```
$ docker pull cfdserver/cfd-server
```

- **Can I use the model to predict certain behaviors?**
  - Yes, but the model is designed to solve a **steady state.** In other words, to reach a stable solution that does not change. A kind of snapshot of the data center. It is not designed to be **transient**. Therefore the model won't perform on anything that involves a change of state, like the evolution of flow when starting/closing a CRAC or the heating/cooling of the room.
- **Why is the air density and viscosity constant?**
  - The air density and viscosity only change a little in the temperatures that are expected in a server room. The hot/cold air movement (buoyancy) is approximated using an equation. That approximation is precise with less than 1% of error for temperature difference of 15 C in air.
- **Does the model compute the turbulence?**
  - Yes. It uses *Reynolds Averaged Stress*.
- **Can the model follow cyclic behavior?**
  - No, the model tries to reach a **steady state**. Other models can compute those, but they need much more computation.
- **Can I add a specific object, like a chair, in the model?**
  - If the object is small and thin, you probably can ignore it. But if it's larger, you can try to approximate it with the "Box" object. If this is still not enough, you can give us a call so that we add this object for you for a small fee.
- **How do I use a "cloud" service to start the server?**
  - **Digital Ocean (Free 200$ -> https://m.do.co/c/1db9426443ca)**
    i. Login/Signup
    ii. Click on Droplets
    iii. Click on Create (Green Button on the top right)
    iv. Choose a Region and a Datacenter (The closest)
    v. Choose "Marketplace" then search for "Docker". Don't forget to click on the result label (which will be selected in blue).
    vi. Choose "CPU-Optimized" in "dedicated CPU"
    vii. Choose a machine (The more processor the quicker)
    viii. Choose a password. (If you plan to use Digital Ocean often, using an SSH Key is much better.)
    ix. Choose a name and click on "Create Droplet"
    x. Use SSH (putty on windows) to connect to the server.
    xi. Follow the instructions of the Section: Starting the Server on Linux Console
  - **Amazon Web Service**
    i. Follow this link for a Ubuntu 22 LTS machine with docker preinstalled:

https://aws.amazon.com/marketplace/pp/prodview-mm6u6prvppowc?sr=0-4&ref_=beagle&applicationId=AWSMPContessa#pdp-usage

ii. Alternatively you can use any application that offers docker on a Linux platform.
iii. Login to AWS and subscribe to the application. It will take a few minutes for the authorizations.
iv. Configure the software, use a 64-bit system.
v. Choose your machine (EC2 Instance) with a minimum of 16GiB. The more CPU, the quicker the model will return. (c5.2xLarge is a good option)
vi. AWS use the key pairing system, follow the instruction to create one (Only needed once)
vii. Click on launch.
viii. Use SSH (putty on windows) to connect to the server.
ix. Follow the instructions of the Section: Starting the Server on Linux Console

- **Microsoft Azur**
  i. Log in your account.
  ii. Go to the marketplace and choose a docker service with ubuntu 22
  iii. Choose your machine with a minimum of 16 GiB of memory.
  iv. Lunch it and connect to it using SSH
  v. Follow the instructions of the Section: Starting the Server on Linux Console