

Computational Fluid Dynamics (CFD) Analysis

What is CFD

CFD analysis is used to effectively simulate the fluid flow and heat flow in a defined zone. The CFD analysis uses finite element methods, computer assisted calculations are performed using given data to describe the fluid flow.

This analysis is widely used in aeronautical industry and automotive industry. The application of CFD analysis can also be found in the construction industry, in many cases it has been used to verify or choose the air diffusion system.

The simulations are done in a defined zone with known parameters in terms of airflow and temperatures.

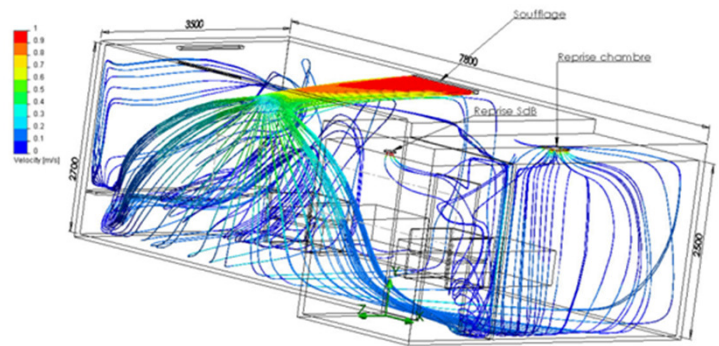


Fig1. CFD Analysis an Hotel Room

Step 1- Define the zone for analysis

Numerical simulation of a zone through CFD analysis helps to get an overview of the air flow and heat flow according to a particular arrangement of air diffusers. The first step to perform a CFD analysis for air distribution is to clearly define the occupied space. For very capacious spaces like malls and airports some critical regions can be studied to facilitate calculations and minimize the time required to complete modeling and analysis.

A 3D modeling of the studied area is realized using a CAD software, this 3D model represents the zone including major obstacles to airflow. In figure 2 the analysis represents a building with 2 floors with a central mechanical stair and an atrium. In this particular case the complete area is not required for analysis as it can be considered with repeating symmetries.

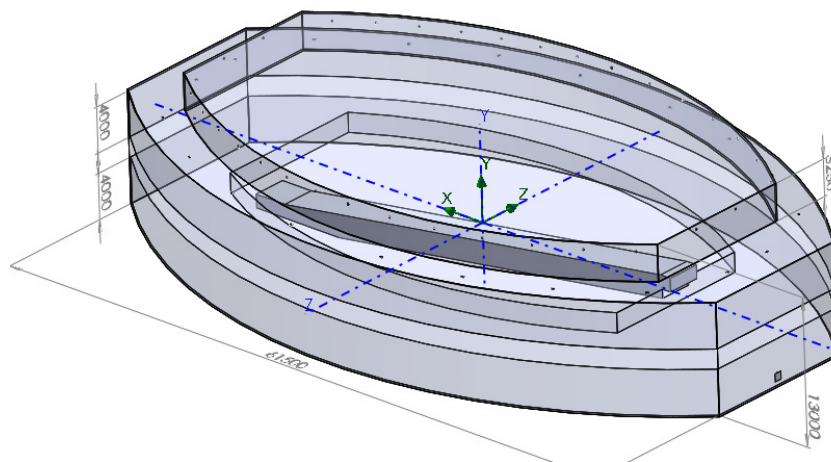


Fig 2. 3D Model of analyzed zone

Computational Fluid Dynamics (CFD) Analysis

Step 2- Define the aeraulic and thermal limits

As mentioned previously, the analysis is done with defined boundaries and not only for the physical analyzed space. The following information are mandatory to perform a proper CFD analysis:

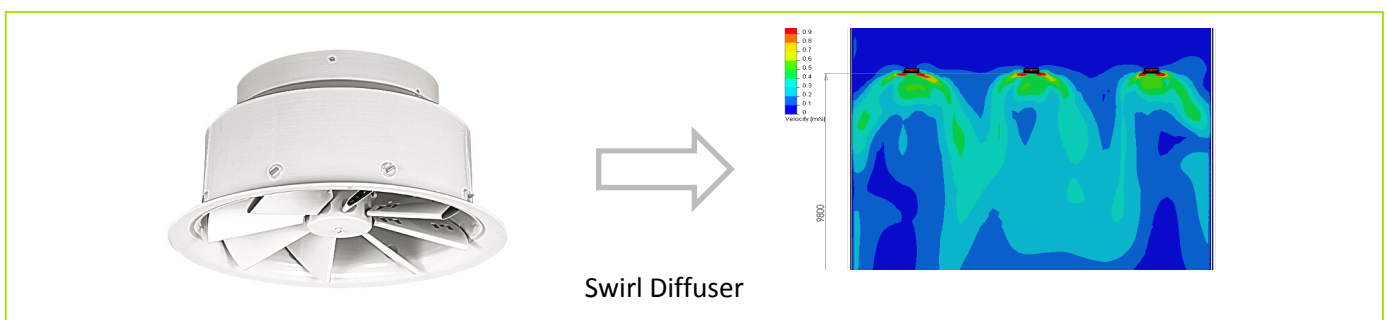
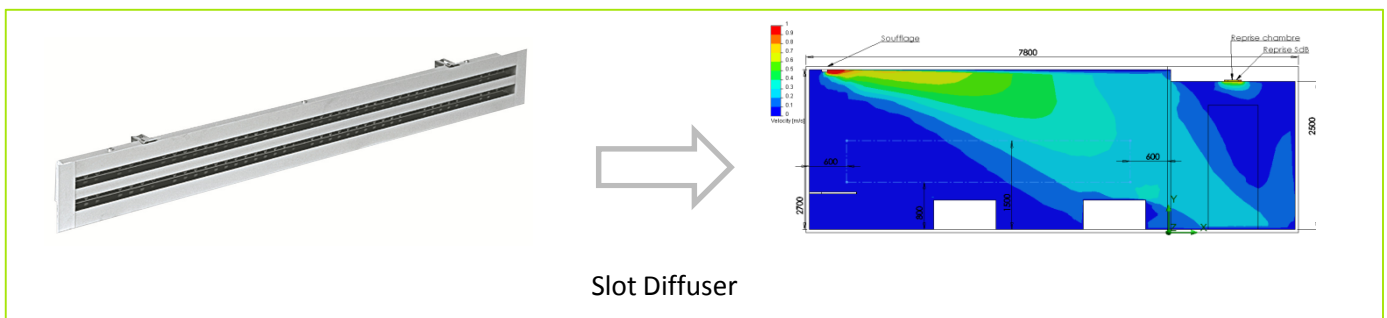
- Airflow per diffuser.
- Temperature of air leaving the outlet.
- Desired temperature inside the room.
- Type and number of diffusers.

	Airflow (m3/h)	T supply (°C)	T indoor (°C)	Supply diffuser	Diffuseur reprise
summer	1440	13	24	AR 883 D400 – 45°	AR 871 D315

Fig3. Table of required data for CFD analysis

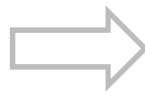
Step 3- Type of Air outlet

Each type of air outlet has a different aeraulic characteristic which clearly defines the type of air flow pattern. For proper CFD analysis the air outlet design should be integrated in the calculation else the simulation will not reflect the proposed installation air and heat flow in the studied zone. The figures below show different airflow patterns according to product type.

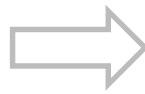
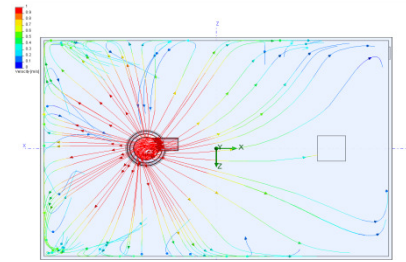


Computational Fluid Dynamics (CFD) Analysis

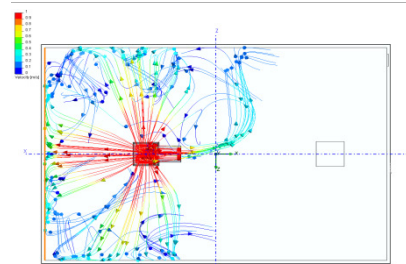
Step 3- Type of Air outlet ... contd.



Twisted Diffuser



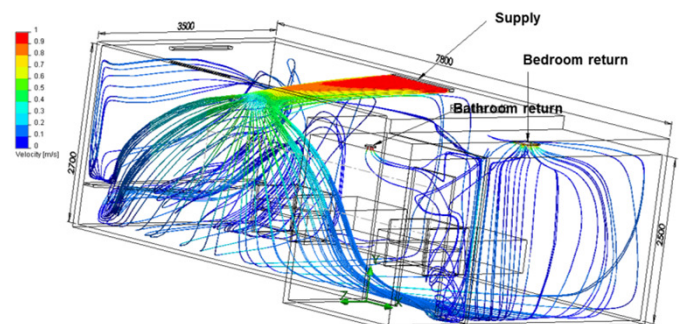
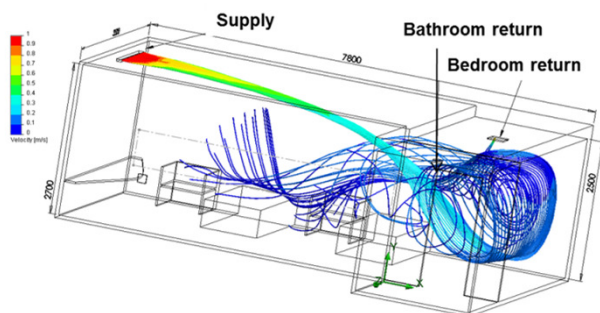
Square Diffuser



Step 4- Position of Supply and Exhaust Diffuser

To finalize the analysis the exact position of each diffuser should be clearly specified. The supply and return diffusers position has a great impact on the air flow pattern. To ensure proper comfort in the occupied space the diffuser should not create regions of high air velocity or improper air mixing in the defined occupied space.

On the figure below, the CFD analysis clearly demonstrate how the position of the supply diffuser changes the air diffusion in the room to ensure better comfort without wasting energy.



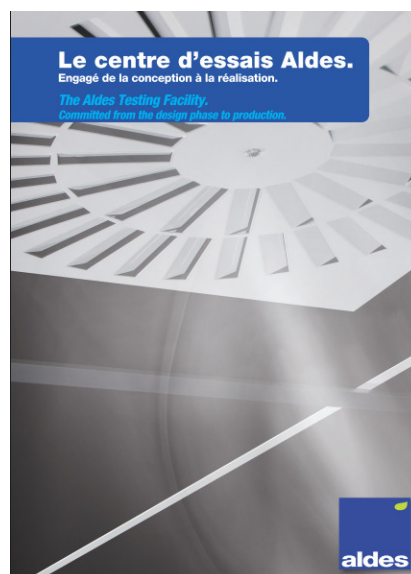
Computational Fluid Dynamics (CFD) Analysis

Aldes Technical Expertise in Air Diffusion

For the past several years Aldes has proven its technical expertise in the air diffusion domain. With a wide range of air diffusers, state of the air Testing Facility in France and considerable innovation in air diffusion, Aldes aims for better thermal comfort in buildings. There are 3 main points to consider when choosing an air diffuser: the length of air throw, the terminal velocity in the occupied space and the sound power level. Considering these main important points Aldes develop new products in line with these criteria and unique aesthetic design.

In the Middle East Region our subsidiary provide with technical support in air diffuser selection. Our team members select the right diffusers according to design constraints and building characteristics and project constraints. Adding to this we propose in specific cases, Computational Fluid Dynamic (CFD) analysis for a better selection and arrangement of air diffusers. The analysis is performed according to given conditions by the client, a 3D model of the studied zone of the building is realized and actual conditions are simulated.

In many projects high quality Air Handling Units are installed but when it comes to the air diffusion in the building, the design is not matching even basic requirements for comfort. Inadequate sizing and selection can cause discomfort in the occupied areas, uneven temperature in the building and sometime over loading of the Air Handling Unit. The selection of air diffusers is to be considered as an important part of HVAC design in a building. Aldes Middle East has performed several CFD analysis for prestigious projects in the region.



For update regarding other products or sending any enquiry,
please visit our website

www.aldes.ae

