

# GADEN TUTORIAL

## FROM CAD MODEL TO GAS DISPERSION

This document is a guide of how to create a new environment for the GADEN gas dispersal simulator. It is written having in mind people with few or non-existent experience with computational fluid dynamics or CAD design, so it is more a “for noobs” guide than anything else.

It is assumed that you have read and understood the working mechanism of GADEN ([link](#)), and that you have downloaded and compiled the GADEN ROS pkgs ([link GADEN repository](#)). For further information please refer to the project web page at: <http://mapir.isa.uma.es/work/gaden>

J. Monroy, V. Hernandez-Bennetts, H. Fan, A. Lilienthal, J. Gonzalez-Jimenez, ***GADEN: A 3D Gas Dispersion Simulator for Mobile Robot Olfaction in Realistic Environments***. MDPI Sensors, vol. 17, no. 7: 1479, pp. 1--16, 2017

## 1. CAD Model

The generation of a CAD model of the environment is the first step on every GADEN simulation. There is a wide range of CAD tools available both online and offline, and for different expertises. Independent of your election, there are a couple of important facts to be considered when generating the CAD model of our new environment:

- a. It must be understood that the CAD model we are going to employ in the following CFD

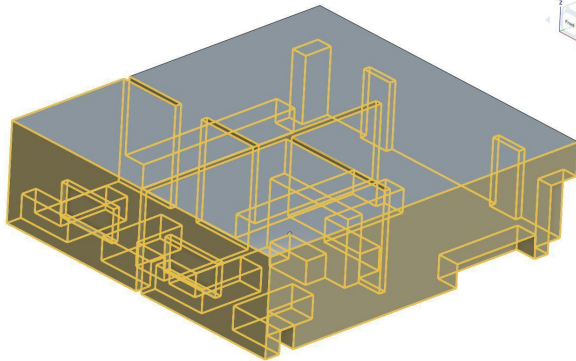
simulation (wind simulation) must represent the “inner” volume of the environment where the gases are going to be simulated. **That is, the input to CFD is not a CAD model of the walls and obstacles of our environment, but just the opposite, the “air” volume within them.** In general this is achieved by creating multiple “parts” in the CAD design, representing walls and obstacles, and then obtaining the inner space as a different CAD part. If you are

not an expert in CAD design (as it is my case), below you can find a very basic guide using a free online tool called **OnShape** (<https://cad.onshape.com>).

- b. Then, it must be stressed that once the CAD model is ready, we need to ensure that **only one CAD part is exported**. That is, during the CAD design, we may have generated different “parts” of our environment (e.g. obstacles, walls, the inner volume, etc), but for the CFD wind simulation (see Section 2) we must only export the inner volume as a single part.

## 1.1. CAD Model Example

In this simple example we employ a free basic online application named **OnShape** (<https://cad.onshape.com>), which has plenty of video tutorials in case of need. As stated before, our



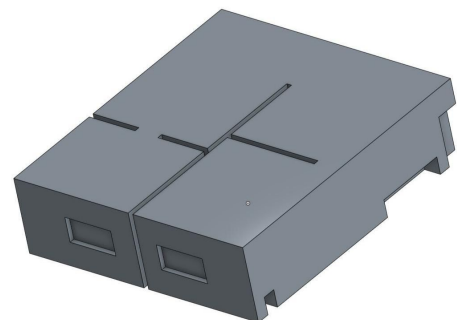
objective is to generate a CAD model of the “inner” of the environment. In the process we may need to generate different CAD parts for the rooms, walls, furniture, obstacles, inlets and outlets, etc., but finally we will use the CAD model of the free space left. Having this in mind, the basic commands to use in the case of

OnShape are:

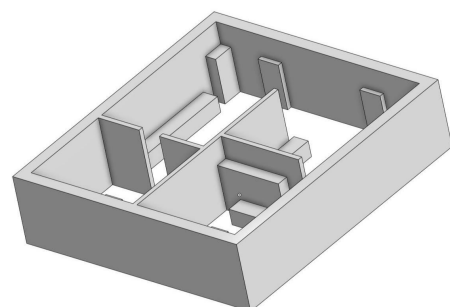
- **Sketch** → Creates a new plane in which to draw parts. It is the basic and mandatory item for every CAD model. We will start with a rectangle (representing the inner volume), and drawing on it any desired obstacle (or we can also use another sketch for it).
- **Extrude** → It allows adding volume to a Sketch. That is, we create a sketch plane, draw for example some obstacles on it, and then rise it up to a volume with the extrude option.
- **Boolean** → This option allows the intersection/add/subtraction of different parts of the CAD model. This is very interesting for merging parts and for obtaining the “inner” volume of the environment as a single CAD part.

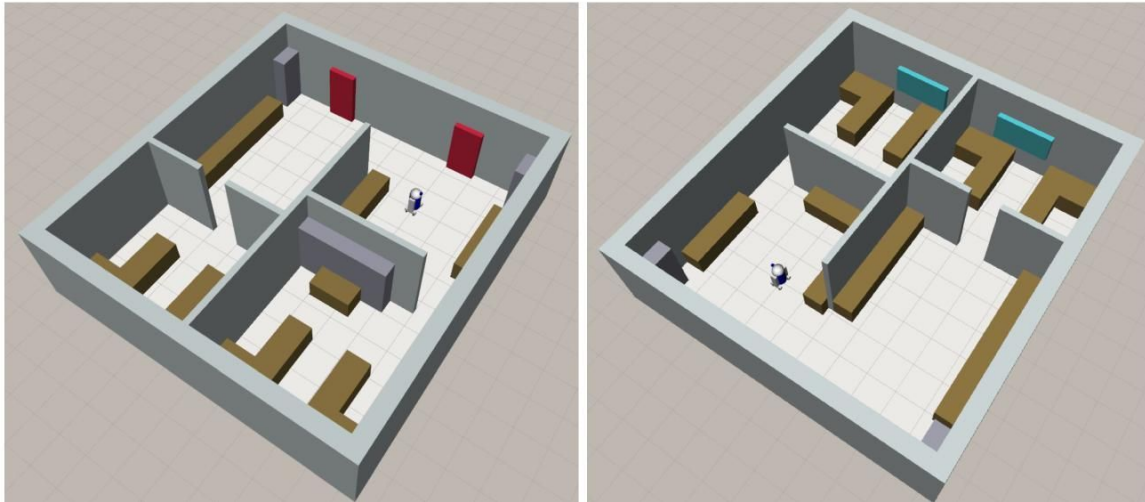
To illustrate this phase, we show here the CAD models for an office-like environment. First, we obtain the CAD model of the inner volume (part\_1), and then a second CAD part representing the obstacles and walls of our environment (part\_2). This second part, not being used for simulation, can be employed for visualization purposes with the “gaden\_environment” pkg.

**PART\_1**: Inner volume where wind flow would be simulated using CFD (see Section 2). This CAD model is composed of only 1 CAD part.



**PART\_2**: The counterpart of the inner volume. It represents the walls, and obstacles of the environment. It is used later in the “gaden\_environment” pkg to visualize the environment in RVIZ. Since every CAD part is represented in RVIZ with a uniform color (no textures), we can split this CAD model in several parts, so each part can be colored differently.





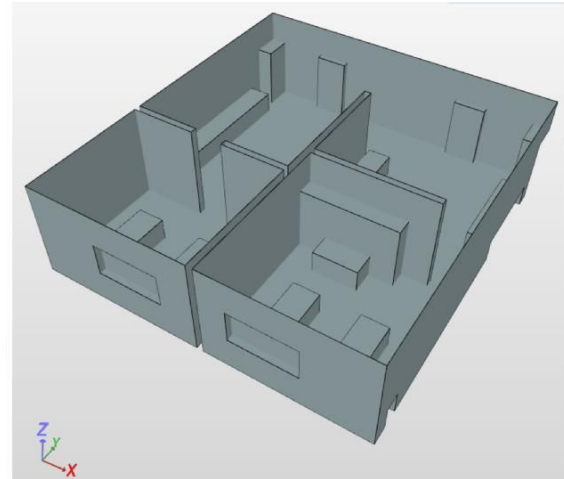
Pictures of the office-like environment as seen on RVIZ. Different CAD parts have been created and colored to improve visualization. All these parts are handled by the “gaden\_environment” pkg.

## 2. CFD Wind Flow Simulation

The simulation of the wind flows within our newly created environment is a very complex process and depends on multiple parameters. The most common approach is to obtain a “more or less” realistic approximation by means of Computational Fluid Dynamics (CFD). In this tutorial we rely on the free and open access platform named **OpenFoam**, arguably one of the most widespread and active open-source CFD projects. OpenFoam takes as input the polyMesh of the inner volume of the environment, together with an extensive list of parameters that control the simulation process, such as boundary wind conditions, transient or steady simulation, turbulence level (i.e., Reynolds number), material properties, solver, etc.

In general, non-experts in fluid dynamics may find this software too complex and with too many parameters, leading to unsuccessful simulations in many occasions. To relax this drawback, different graphical tools have appeared in the last years to control OpenFoam from a more user-friendly interface. This is the case, for example, of **SimScale** (<https://www.simscale.com>), a web-based tool which allows importing the inner volume of the environment as a CAD model, generating the mesh required by OpenFOAM, and configuring the most important parameters of the CFD simulation through an easy and intuitive interface. There are many other tools for interfacing OpenFoam, or you can even work directly with it if u feel confident.

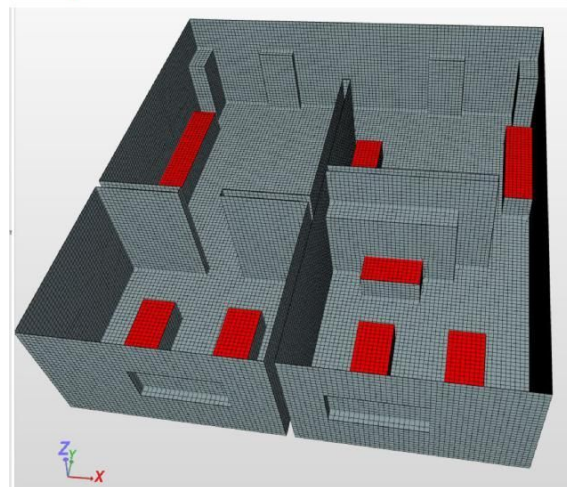
- a) **Import CAD model to SimScale** (only the inner volume part). This simplifies enormously the configuration of the environment in OpenFoam. There are different tools that allow you to create the polyMesh that openFoam requires from a CAD model, here we employ the one integrated in the SimScale platform (see import options).



- b) **Generate the PolyMesh** from the CAD model. It is recommended not to use the automatic mesh tool, but to try to obtain a

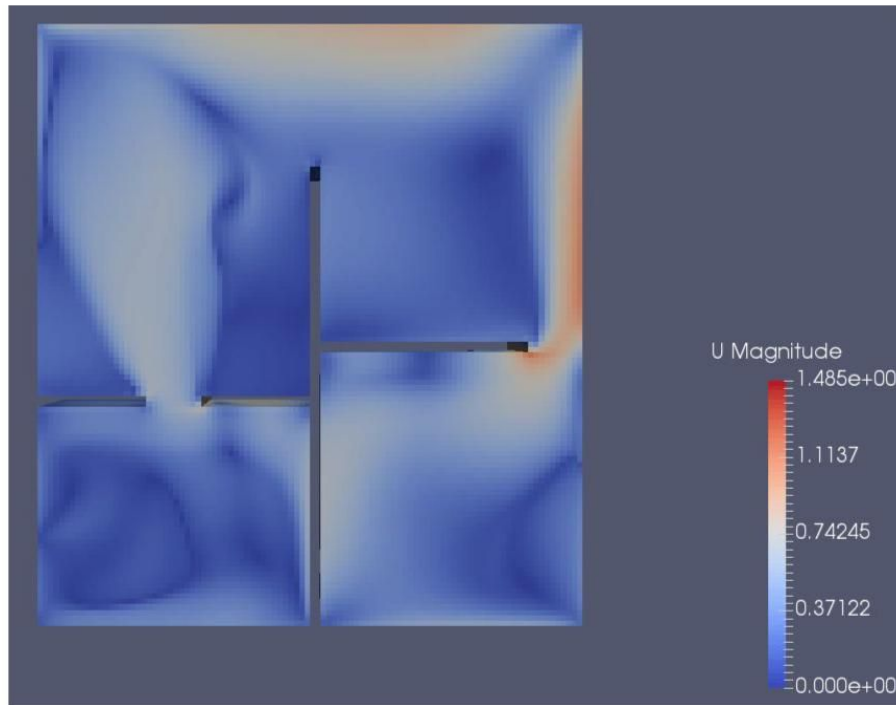
more rigid grid based on cubes by using the “Hex-dominant parametric” tool, under the “Mesh-Creator” tab. The reason is that GADEN will work with a perfect 3D grid of cube cells, so as close the simulations of openFoam are to this perfect grid, the better results we will obtain.

For the “Hex-dominant parametric” tool we need to define the outer environment dimensions (i.e the bounding



box). Since we are only interested in wind simulations within the environment, we can set it to the real dimensions of the environment. Similarly we need to set the number of cells on each dimension. Here we need to ensure that the environment dimensions are multiple of the cell dimension used, since in other case we will face problems with cells falling half in an obstacle, half in free space.

- c) **Run Simulation:** Once the mesh has been generated, we need to set the simulation parameters. We focus on Fluid Dynamics - Incompressible flows - k-epsilon turbulence model - Transient simulation with the PIMPLE solver. Other options can also be selected according to the desired results. We select the mesh to simulate over it, and set the medium as “AIR”. We also need to set the Boundary conditions, setting an input wind speed for the inlets of our environment, and constant pressure at the outlets. The walls and obstacles need also to be defined as “slip/non-slip” boundary conditions. As a tip, set a short interval simulation step to allow a nice convergence of the algorithms, but allow an “adjustable timestep”, setting the max courant number to 0.7. Since this option will cause the time-steps to vary on length, set the writing parameters to “Adjustable at runtime” and select the time interval to save the data.

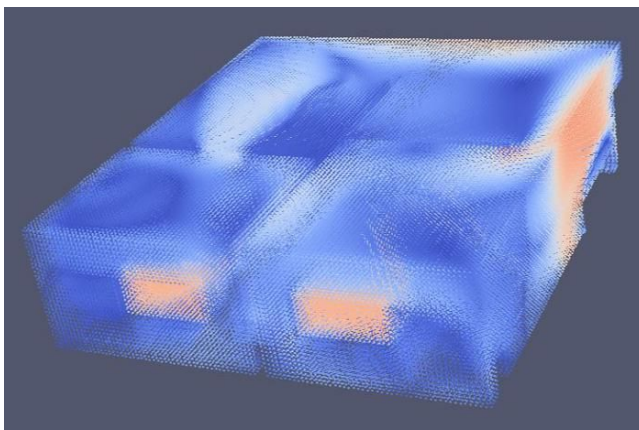


Snapshot of the wind flow in the simulated environment

### 3. PostProcessing of Wind Flow Data

Once the CFD simulation is finished, download the results data (in OpenFoam format) to your computer. Then open it with the OpenFoam viewer, called **ParaView**. With this application you can ensure that the wind flow results are correct (at least they seem to be representative of the boundary conditions you established).

It must be noticed that for most simulations not all cells are perfect cubes, nor they are equally sized. To solve this problem (in GADEN we need a perfect 3D grid with uniform cell sizes), we can apply in ParaView the **“cell centers” filter**, and then **export the data to a CSV file**. This file will be a list of points (x,y,z) with their corresponding wind conditions (u,v,w). This way we simplify the problem of non-uniform cell dimensions. As a tip, when applying the “centerCells” filter, ensure that you have selected the option **“Vertex Cells”** to be able to display the center of the cells as points in the Paraview visualization frame.



Snapshot of the wind flow in our new environment after applying the cell\_centers filter in ParaView.



## 4. The Preprocessing node

At this step, we have all the data we need to start the simulation, but we still need to get it to the right format so that we can feed it to the simulator. That is, we need to turn the CAD models and the point cloud of wind information into cubic grids (of occupancy and wind, respectively).

The **preprocessing** node, included in the simulator, does this work for us. We will need to provide the **preprocessing** node with:

- Any number  $n$  (specified in the parameter “number\_of\_models”) of CAD models (paths specified as parameters “model\_0” to “model\_n”) for the environment in .stl format (if you can’t directly export them to that format, there are plenty of converters online) .
- A separate CAD model (.stl) for the outlets (parameter “outlets\_model”).
- Desired cell size (parameter “cell\_size”).
- The wind information that we got from ParaView. If you have different wind information for each time instant, the node expects you to name them [path]\_i.csv, where [path] is the value of the “wind\_files” parameter, and  $i$  is the instant.
- The coordinates of point of the map we know is free (not inside of an obstacle). This is used to differentiate free space from the interior of obstacles. (Parameters “empty\_point\_x”, “empty\_point\_y” and “empty\_point\_z”).

After the node runs, we will have:

- A 3D occupancy grid file where each cell is 0 if free, 1 if occupied or 2 if it’s part of an outlet.
- Three wind grid files for each time instant we originally had information of. Each of the files contains one of the components (U, V, W) of the wind for each cell.
- A 2D map of the environment. Note that this is a XY plane, so if your models are rotated with the Y axis pointing upwards, this will look weird.

## 5. GADEN

Once we have the CAD models of the environment, the CFD wind flow simulations, and the Occupancy 3D map, we can start working with GADEN to obtain the desired gas dispersal simulations. For a description of the different packages and implemented functionalities we refer the reader to the official article where GADEN was presented ([link here](#)).

Also, we have included in the github repository a “gaden\_demo” directory with launch files for the different pkgs where all the parameters are described in detail, and a small program to decompress and make human-readable the log files generated by the filament simulator. For further information please refer to the project web page at: <http://mapir.isa.uma.es/work/gaden>