Complete Tutorial to Generate a New Scenario on GADEN

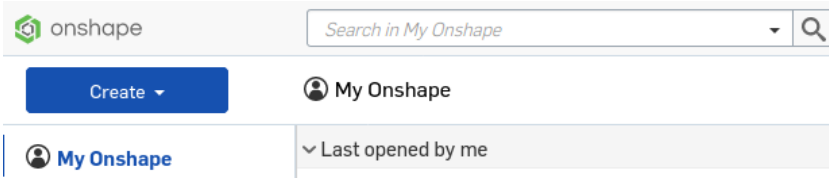
by Maurício Souza Sathler and Héctor Azpúrua

OnShape

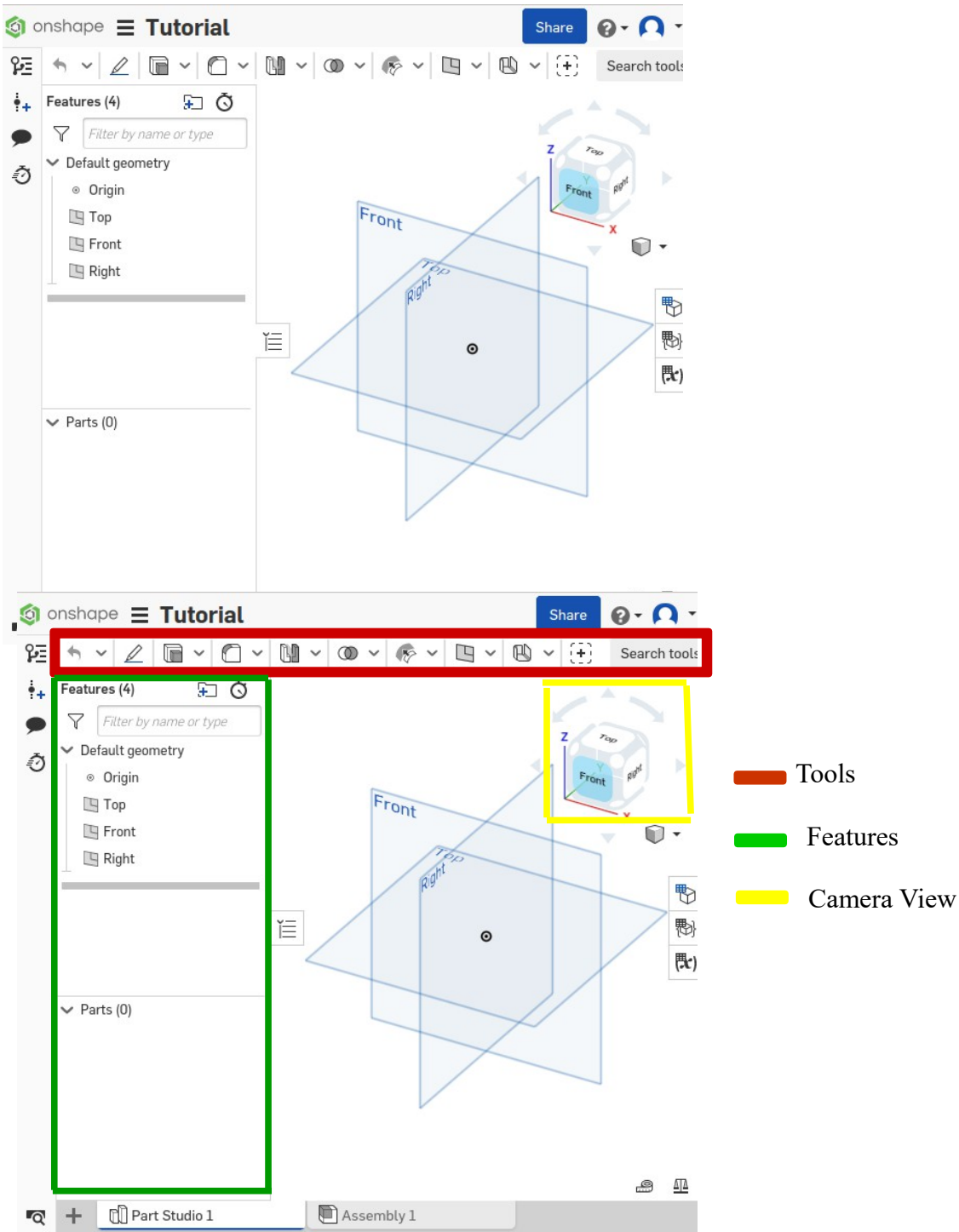
How to create an environment in GADEN simulator.

Generate Mesh onShape (https://cad.onshape.com/):

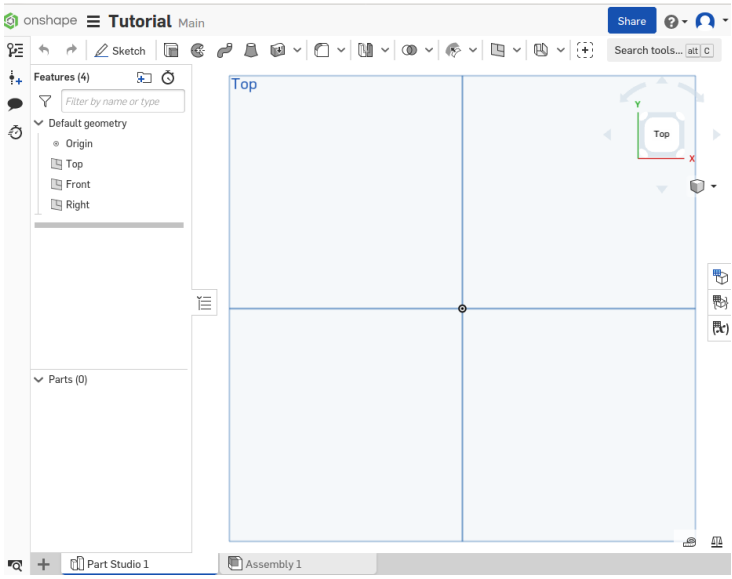
Create new document:



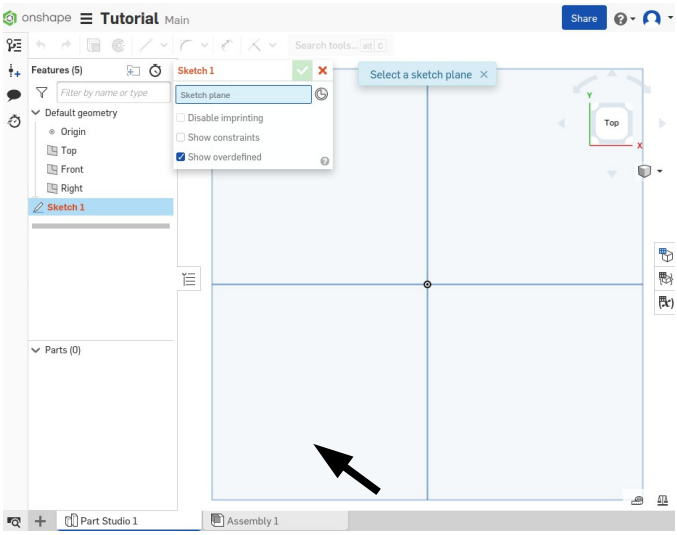
Empty document:



1 - Left click in Top (Camera View):

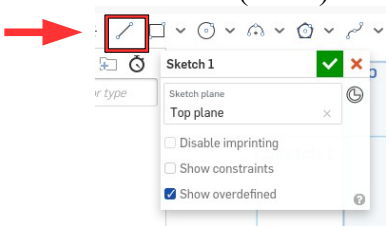


2 – Select Sketch (Tools):

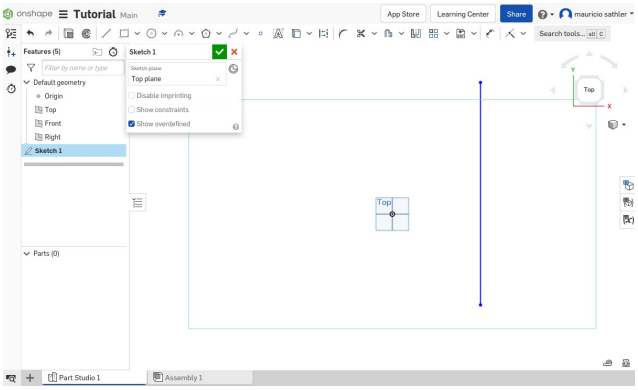


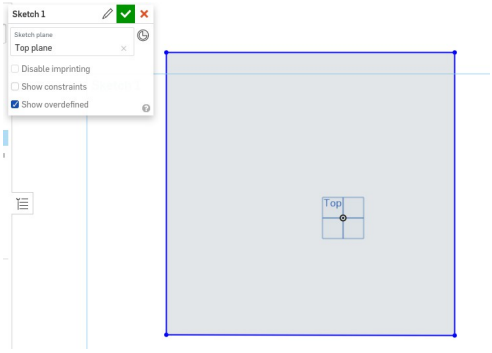
The tool wait what plane the sketch will be write, so left click on top plane.

3- Select Line (Tools)

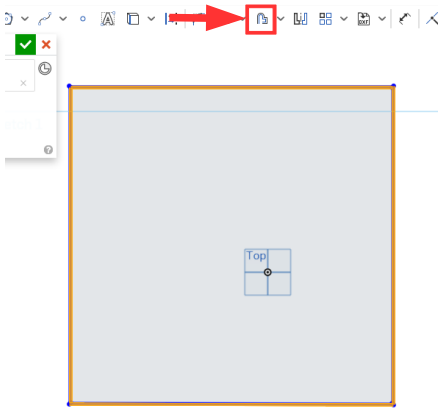


Scroll mouse to remove zoom to better vision and draw a square:

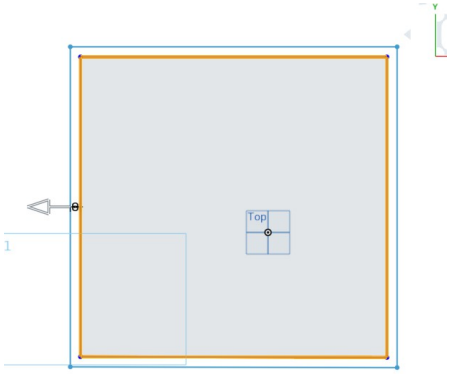




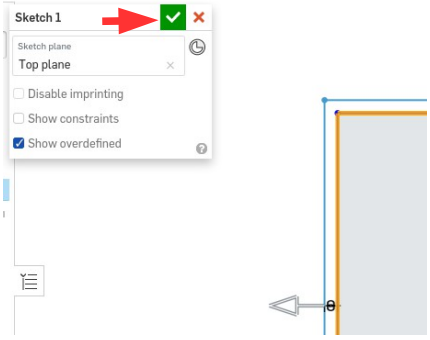
Left Click on the lines after click offset (Tools):

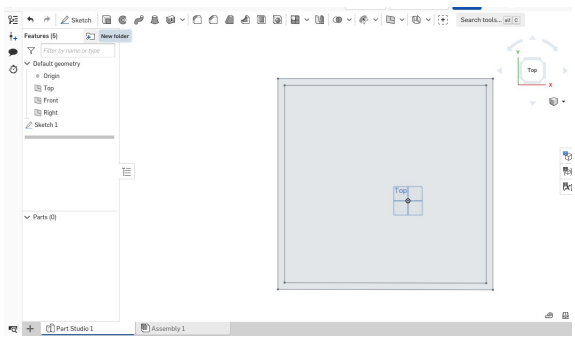


Pull the arrow like img bellow:

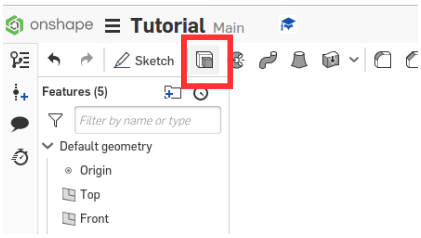


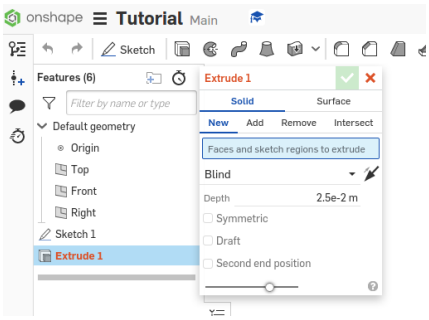
Finish the Sketch 1 “V”



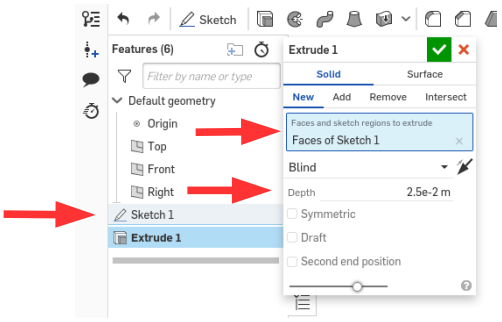


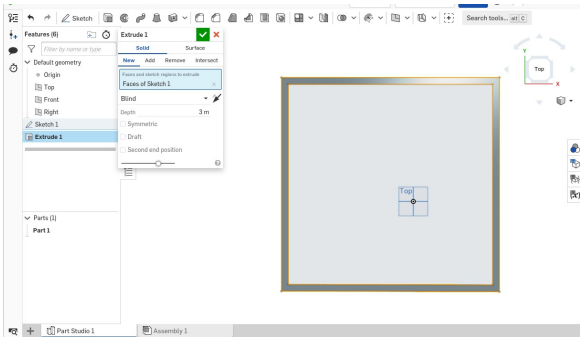
Now Select EXTRUDE (Tools)

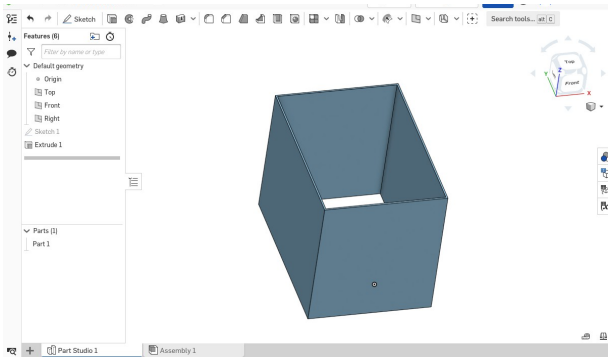




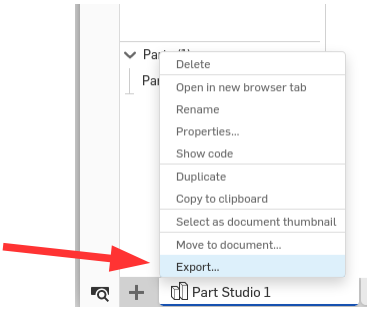
You must select the faces to use the tool, then left-click on Sketch 1 after this write 3 into Depth box:

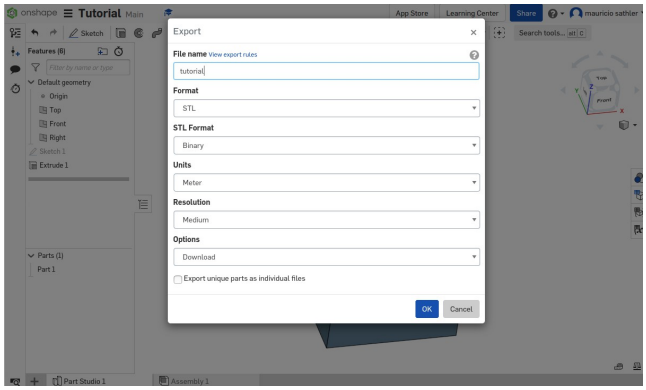




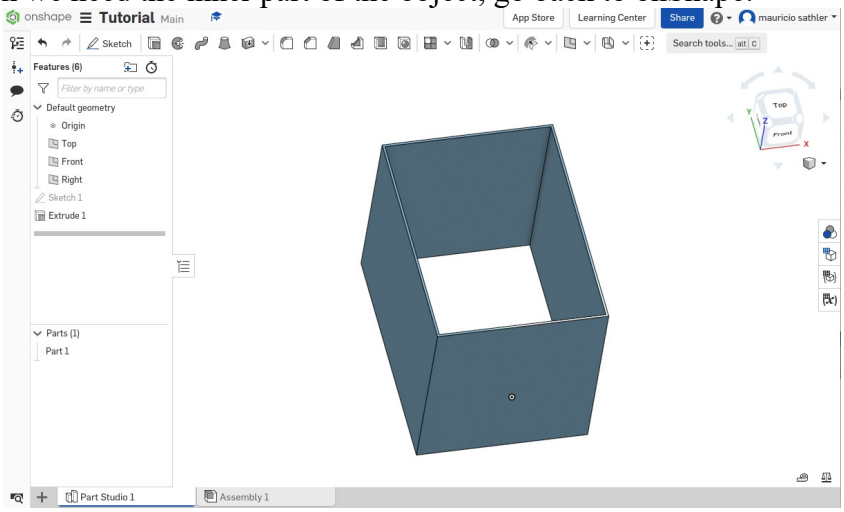


Export Files, right-click on **Part Studio** 1, then **Export…** select your **File name**, **Format** to STL, **STL Format** to Binary and **OK**:

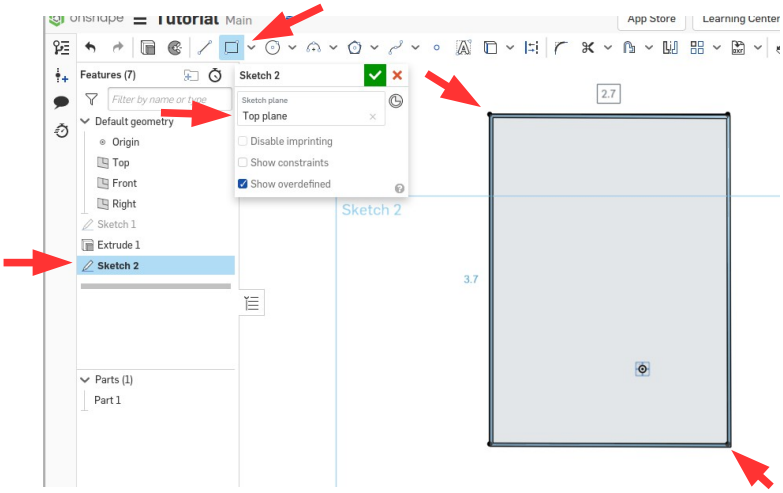




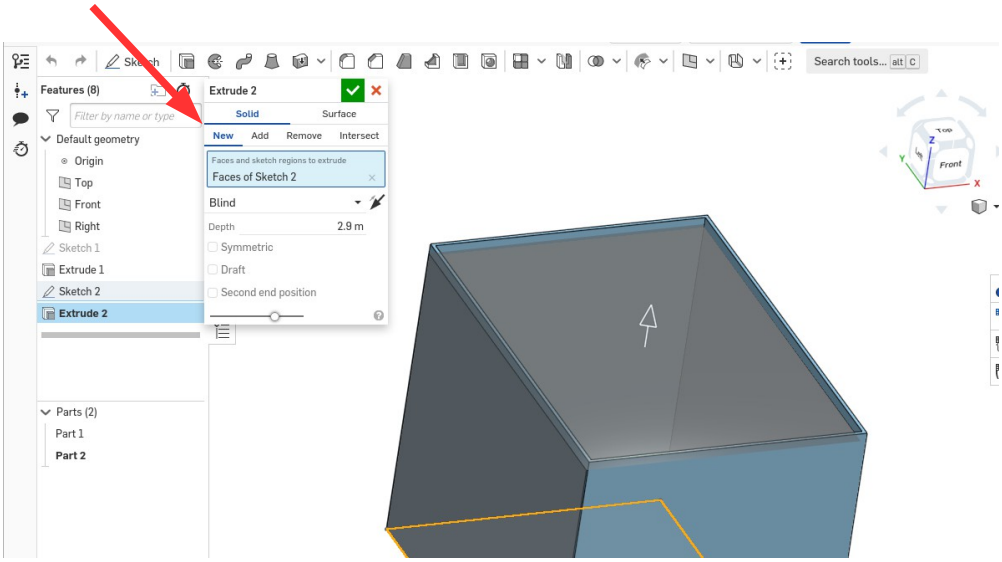
Now to simulation we need the inner part of the object, go back to onShape:



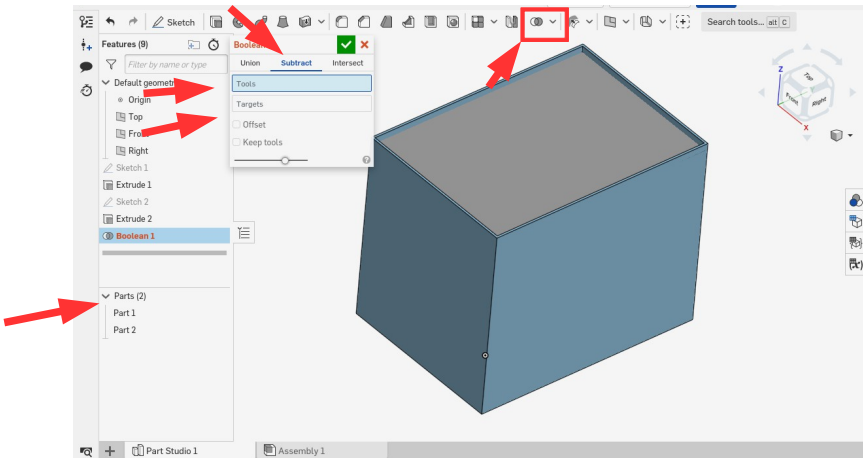
Select Sketch (Tools) and Sketch plane → Top:
Corner Retangle (Tools) then left-click on the upper left part of the drawing and then on the lower right part.
Click **V**

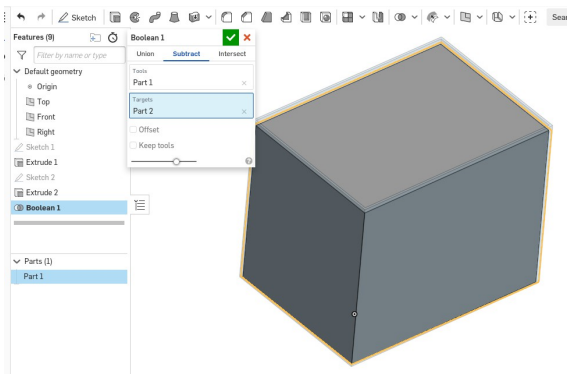


Now, extrude the Sketch 2 with 2.9m of Defth:
Obs: Its a new part.



Now click on Boolean (Tools)
- Substract
- Tools → left-click on Part 1
- Targets → left-click on Part 2
- Click **V**





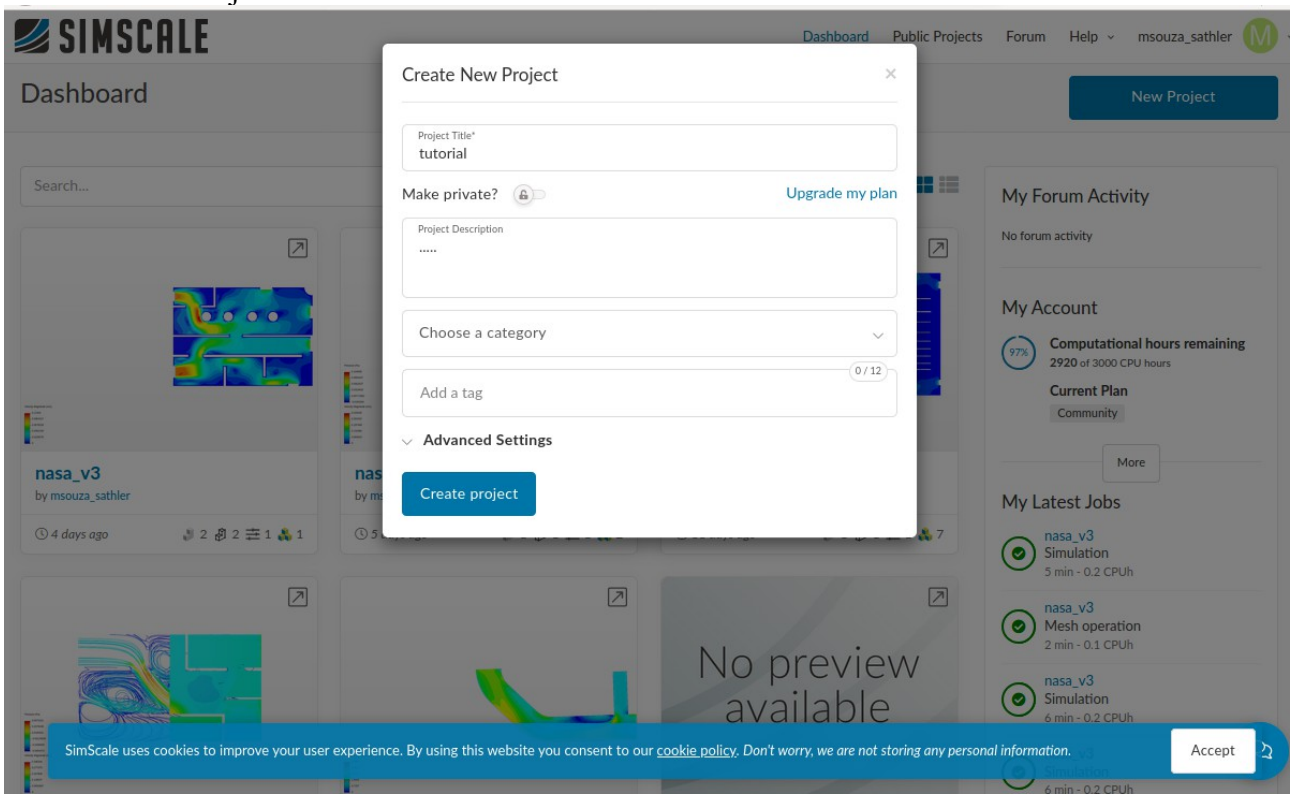
Export inner part, right-click on **Part Studio** 1, then **Export…** select your **File name**, **Format** to STL, **STL Format** to Binary and **OK**:

SimScale

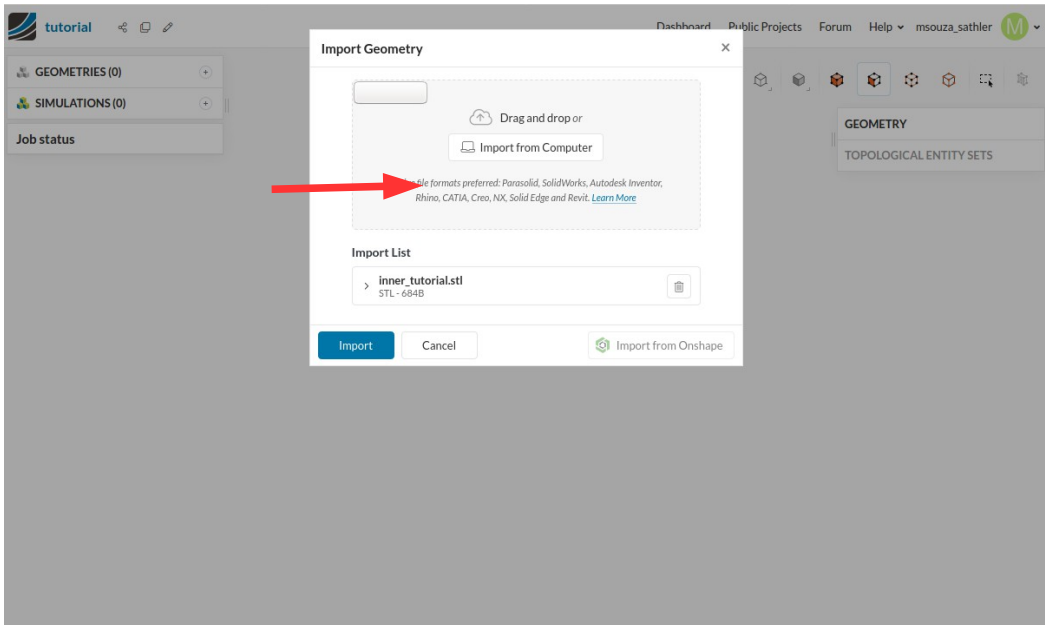
https://www.simscale.com/

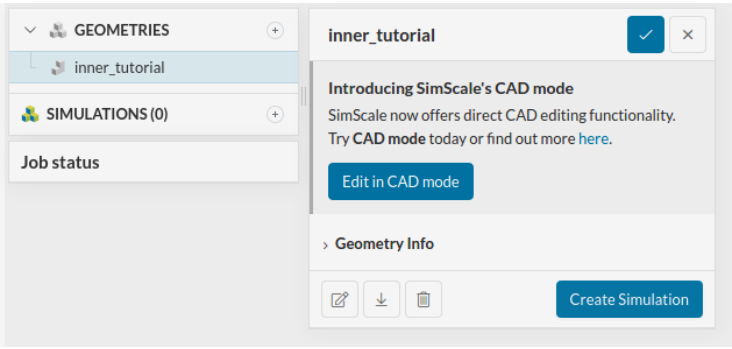
For this tutorial we will use a more complex structure for better visualization.

Create a New Project

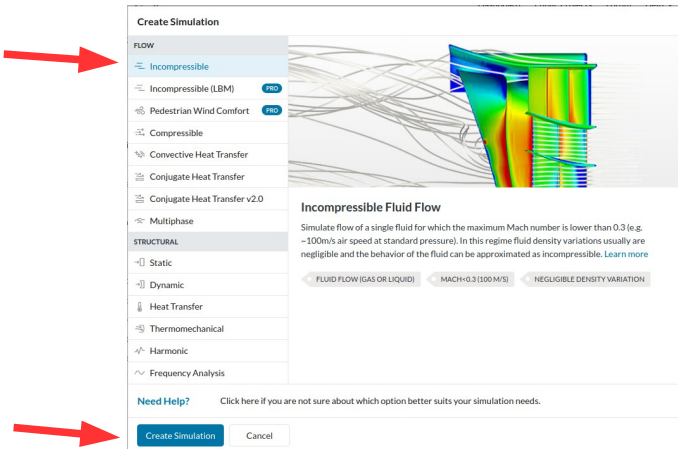


Import Geometry:
- Import the inner stl





Create Simulation → Incompressible → Create simulation



Steps needed to set up the simulation:

1 – Mesh

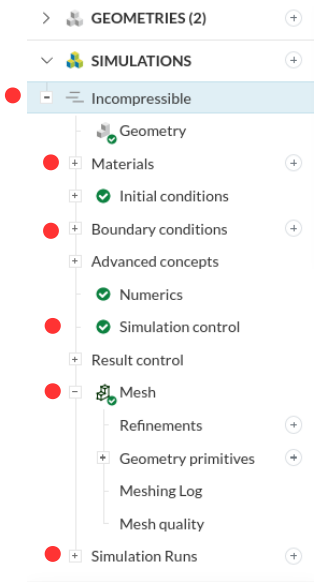
2 – Incompressible

3 – Materials

4 - Boundary conditions

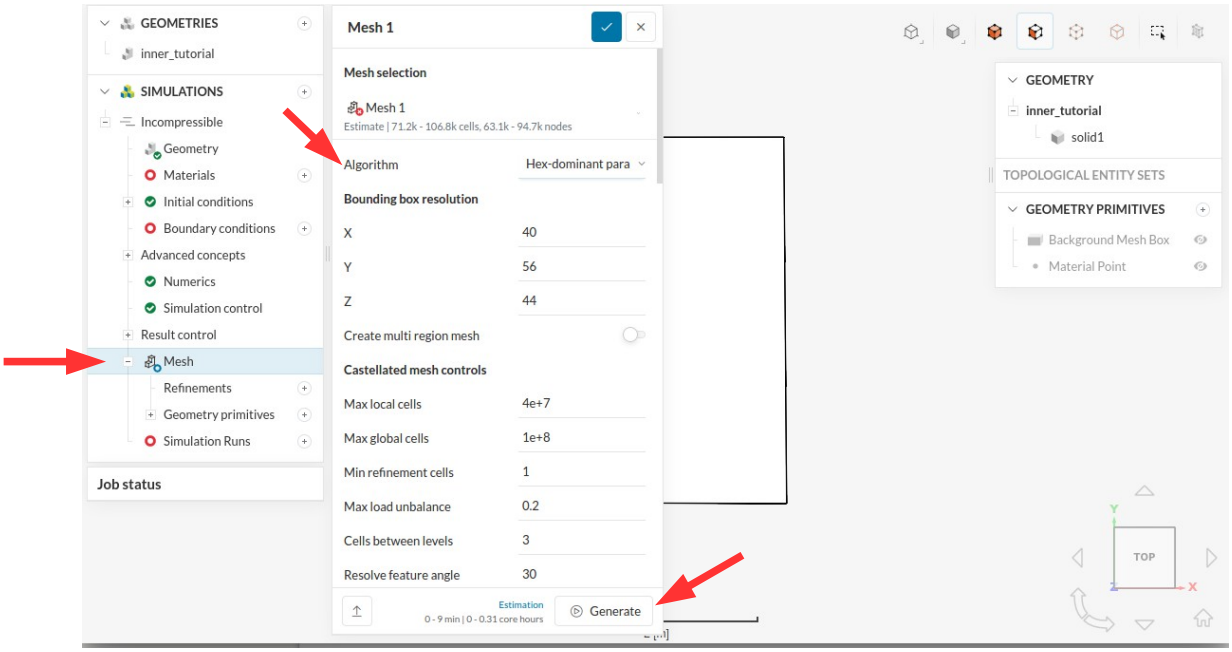
5 - Simulation control

6 – Simulation Runs

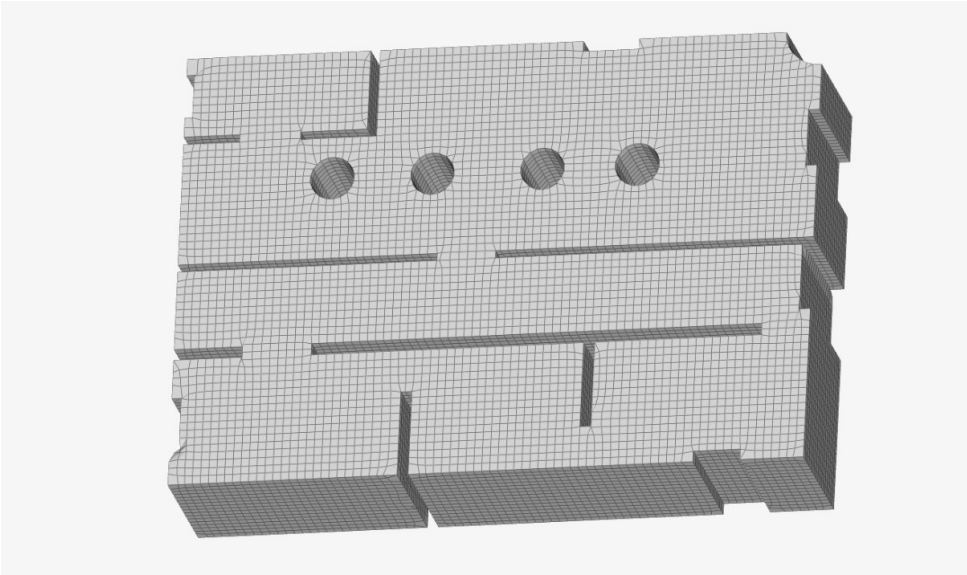


1 - Mesh:

- Set Algorithm to Hex-dominant parametric, then Generate:



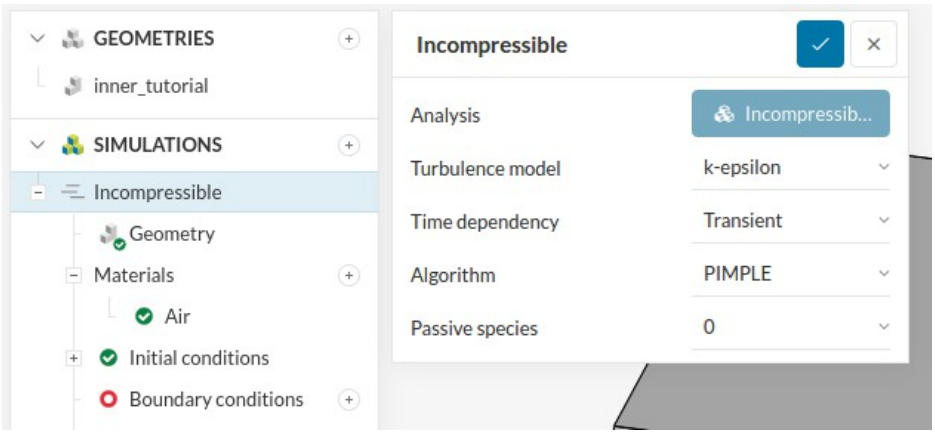
In the same place as the Generate button you can track the time left to generate. After finishing you should get something like this:



2 – Incompressible

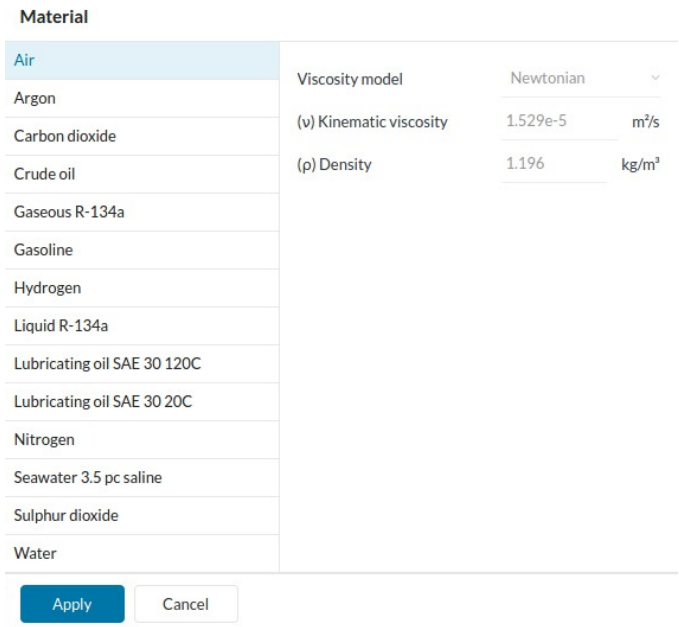
Set:

- Turbulence model → k-epsilon
- Time dependency → Transient

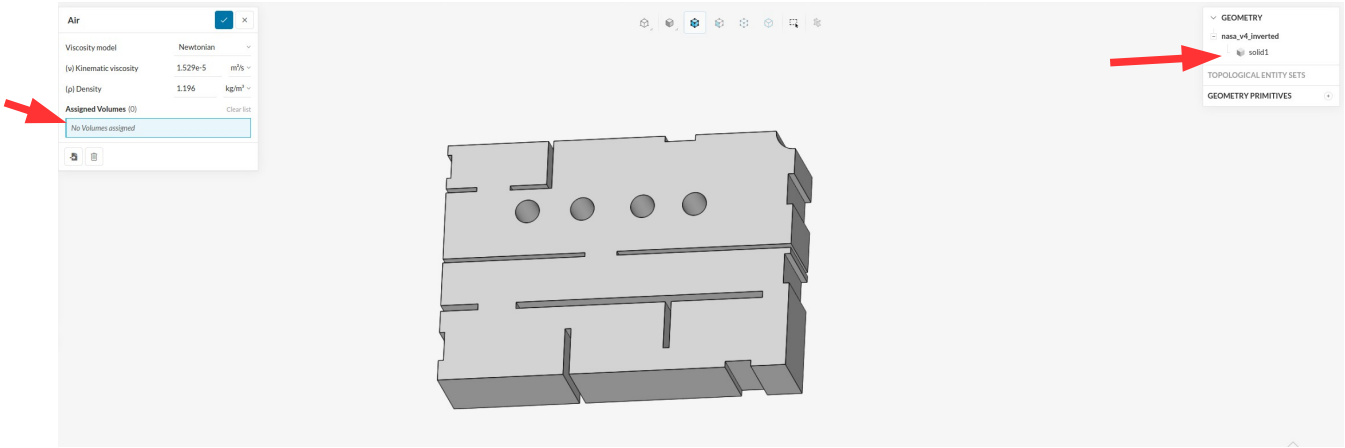


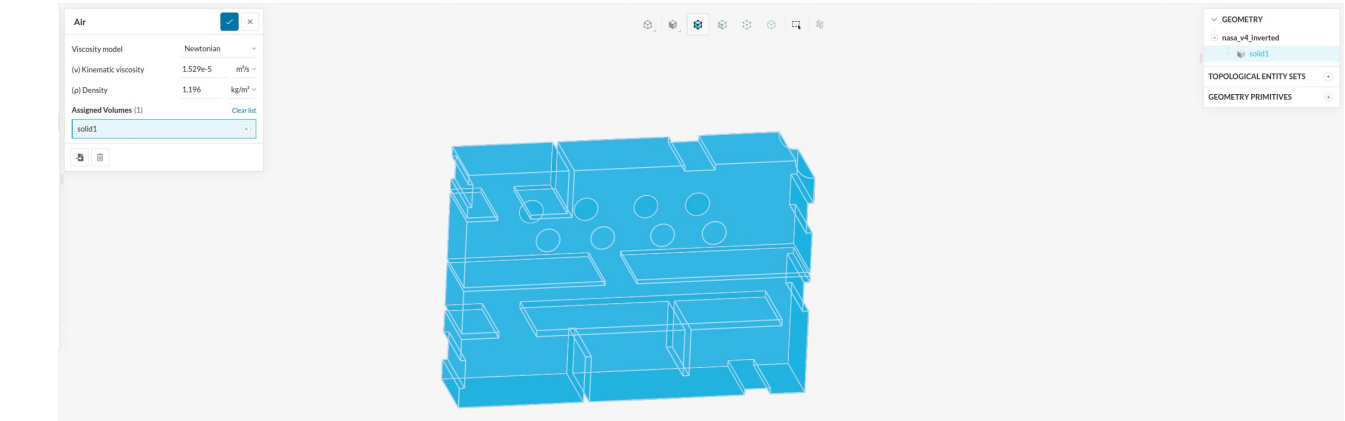
3 – Materials

Select Air, then Apply



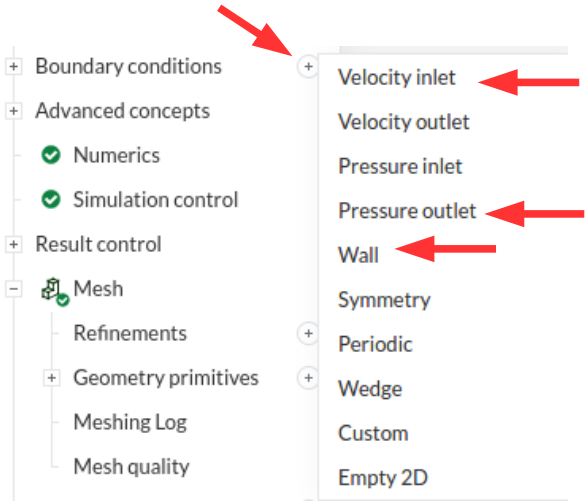
It is necessary to select where the air is present, so select the solid



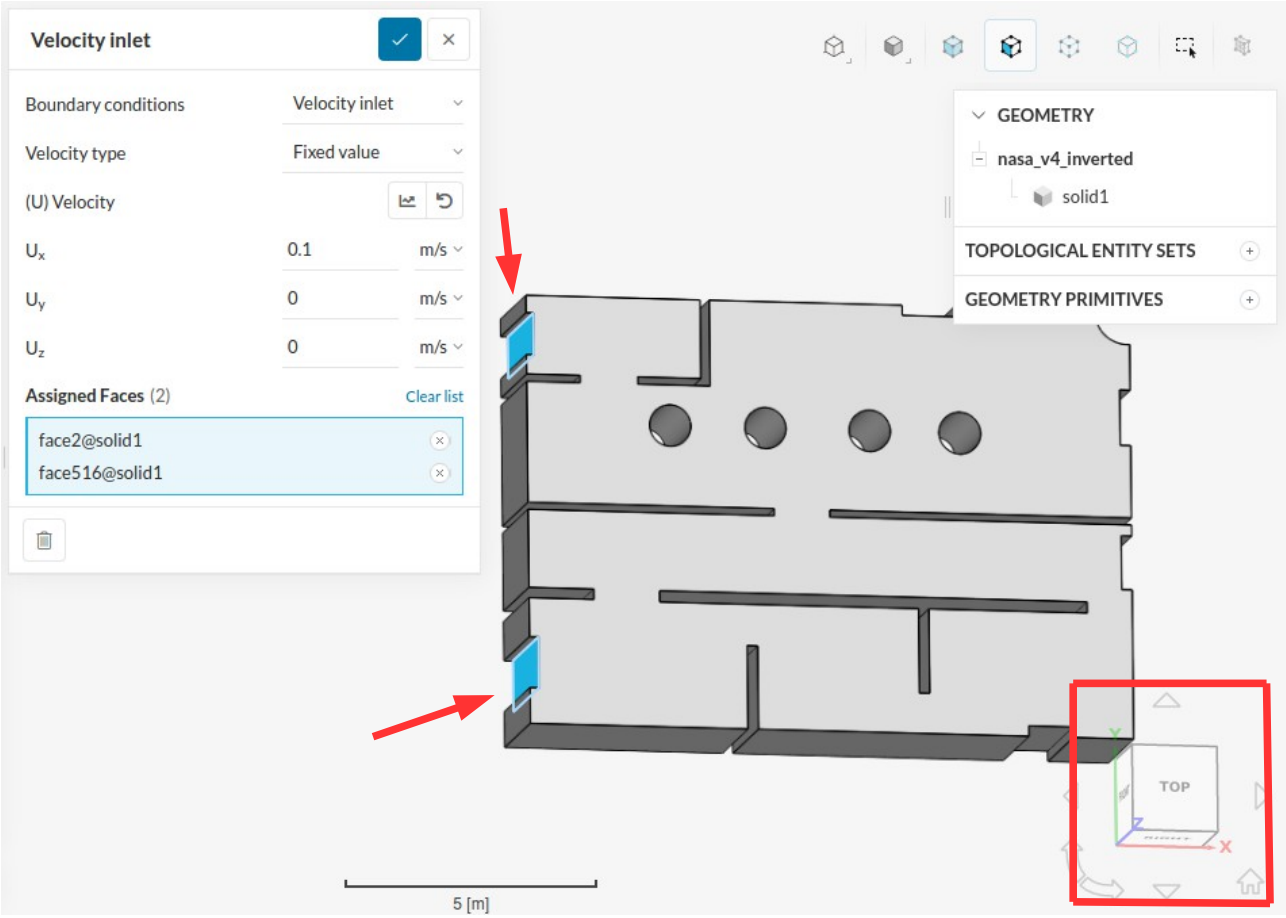


4 - Boundary conditions

By Selecting **(+)** will be appear some configurations, but for our simulation we need only three:
- Velocity Inlet: Is the inlet wind speed.
- Pressure Outlet: These are the air vents.
- Wall.

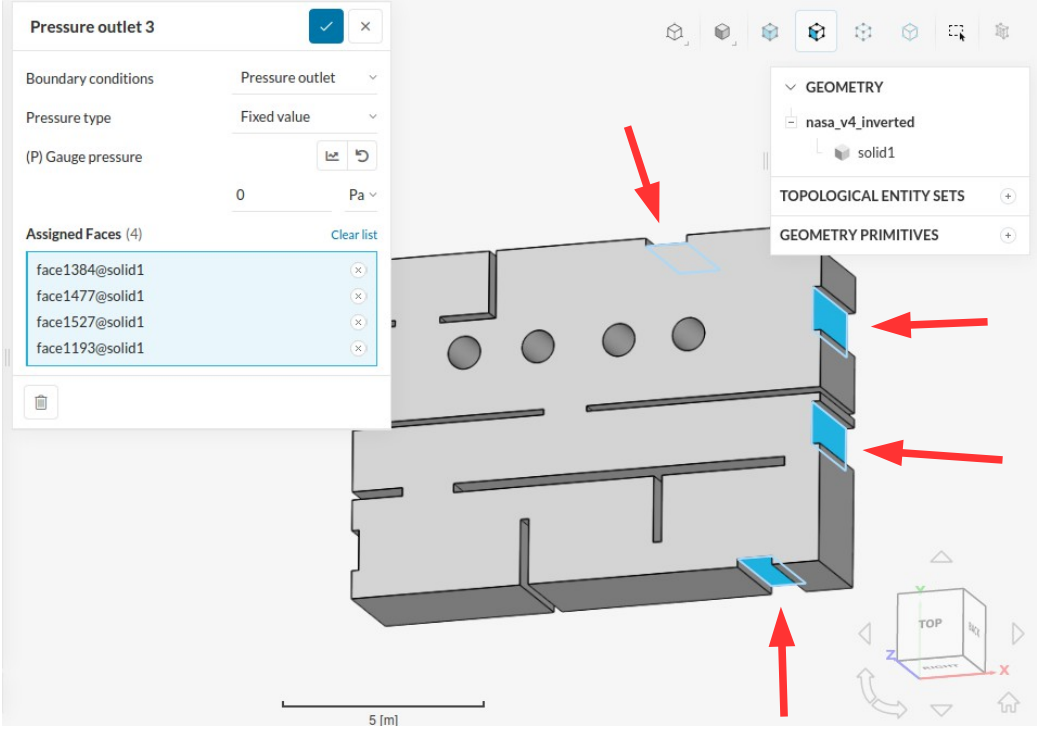


- <u>Velocity Inlet:</u>
Lets select our faces of wind inlet, in our case the speed has to be on the **X** axis because it doesn't make sense on another axis for these windows (Obs: To define this always look at the coordinates)
Set the wind Velocity as 0.1m/s



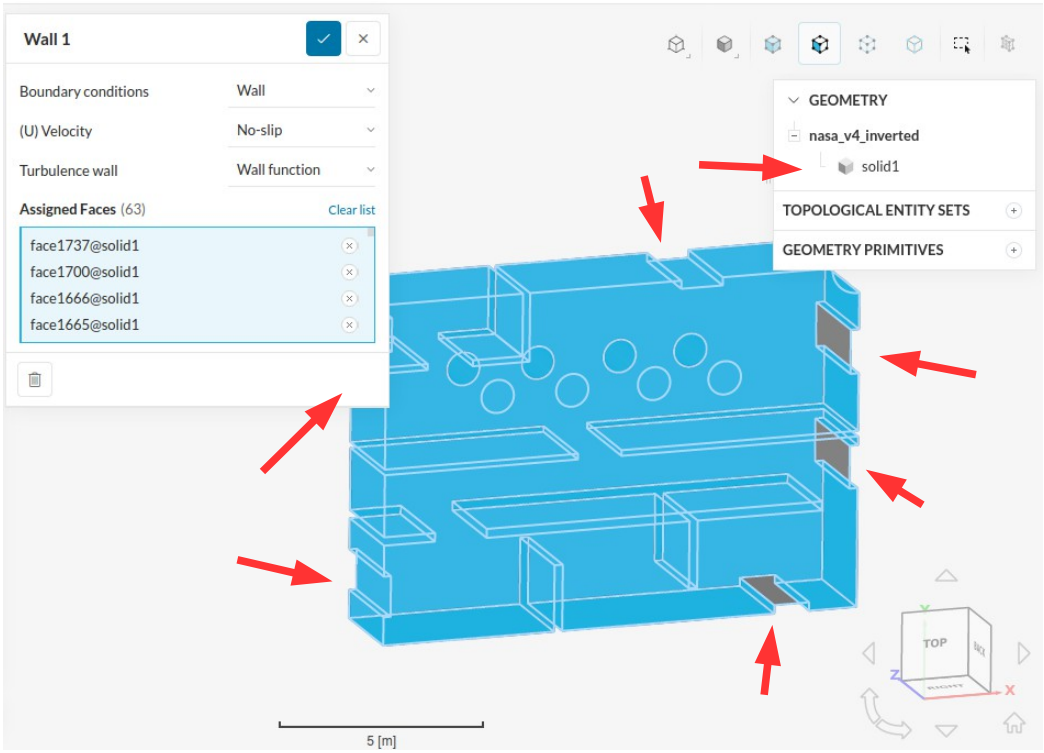
- Pressure Outlet:

Only need to select the faces, like the image bellow:

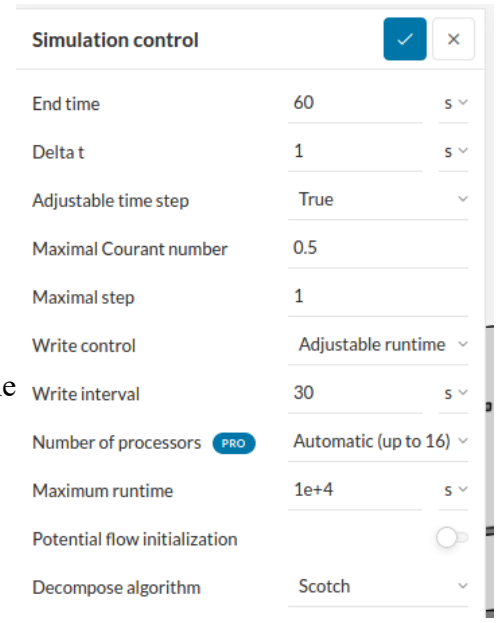


- Wall:

Select the rest of the faces removing the previously selected ones. To reduce effort it is interesting to select the whole solid and deselect the air inlets and outlets



5 - Simulation control



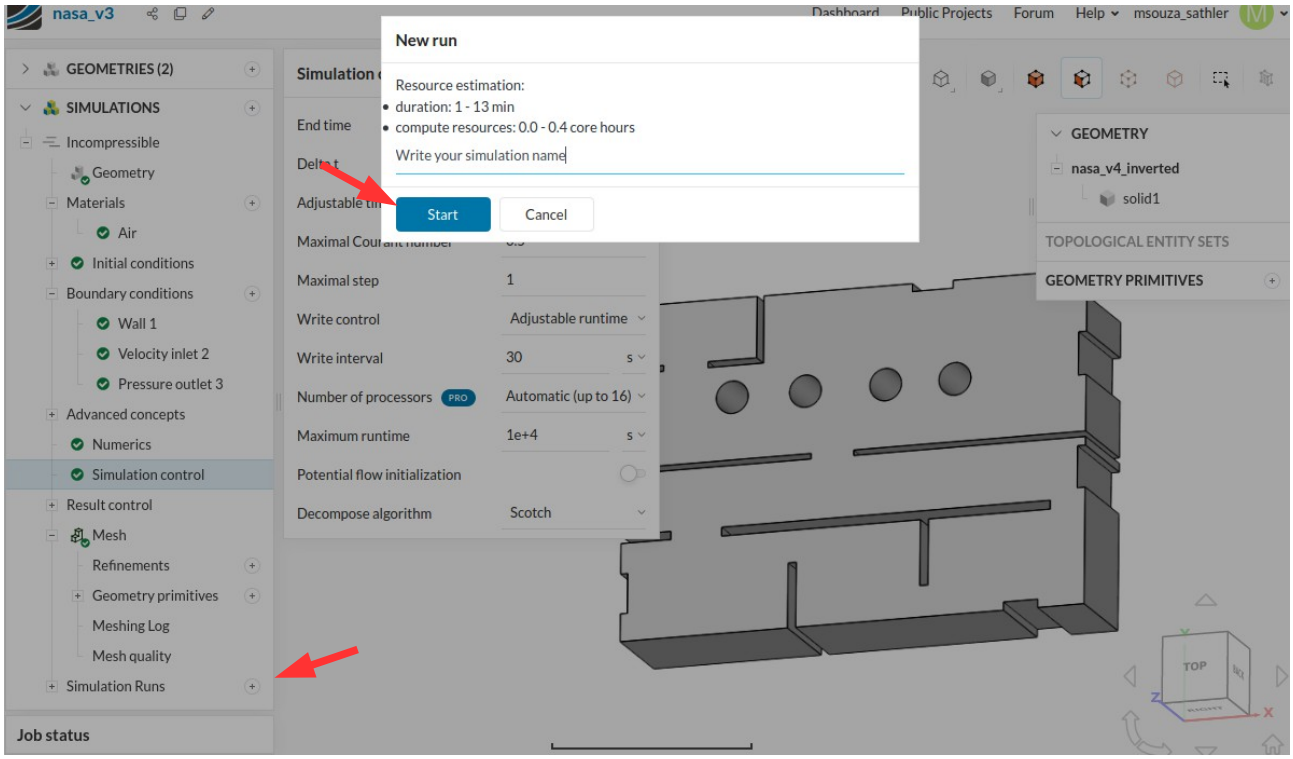
Configure the settings as shown in the picture to the right.

If you want to change something remmember of important points:
- Maximal **Courant number** have a interval of 0.5 to 0.7
- Write interval has to be divisible from the **End time**
- The shorter the W**rite interval** and **Delta t** the longer the time to simulate.
- The longer the **End time**, more time it will take to simulate
- If you don't know how to set the parameters well, don't change the write control from Adjustable runtime.

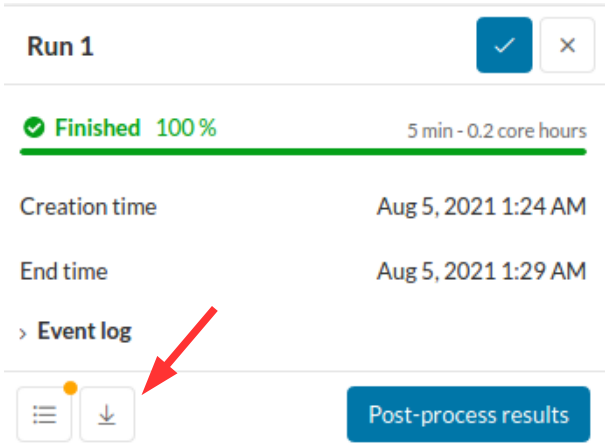
After these steps we are ready to start the simulation!!

6 – Simulation Runs

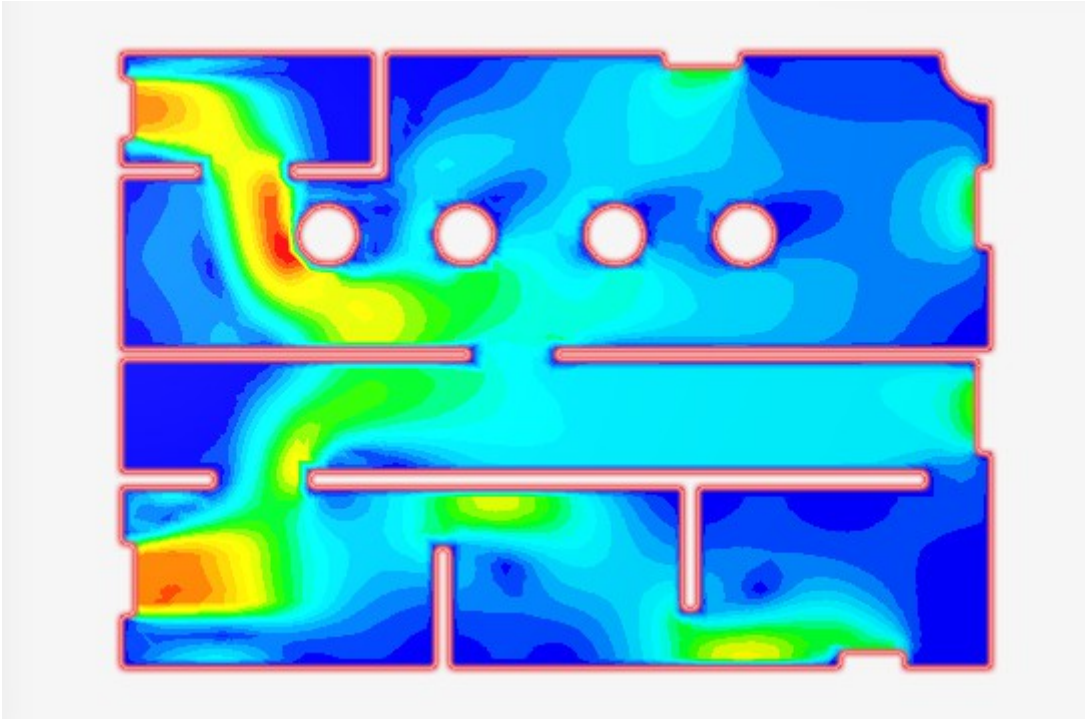
Start your simulation and wait. After finish the simulation.



When the simulation finish download the simulation file.
If you want to see the results online click in Post-precess results.



Simulation result:



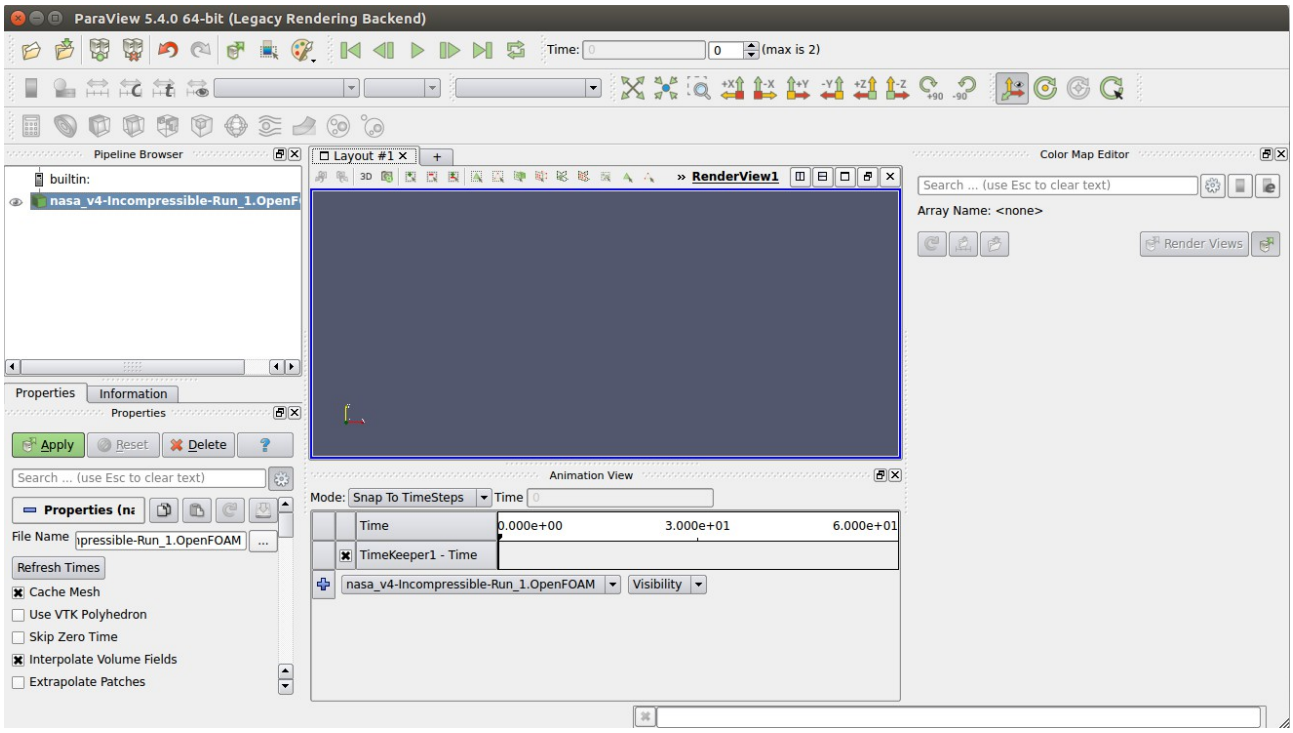
OpenFOAM

Extract your compressed files.

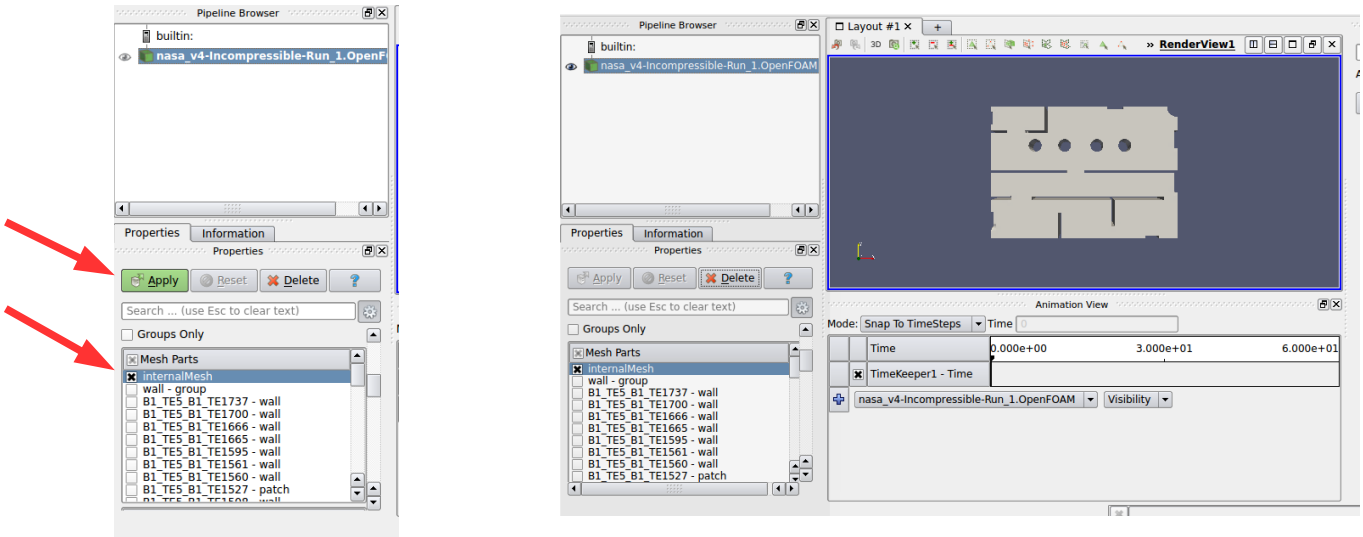
Open the terminal and go to the extracted file, type paraFoam.

```
itv@nt005:~/Desktop/nasa/STL/nasa_v4/nasa_v4-Incompressible-Run_
1$ ls
0  30  60  case.foam  constant  nasa_v4_0.csv  system
itv@nt005:~/Desktop/nasa/STL/nasa_v4/nasa_v4-Incompressible-Run_
1$ paraFoam
```

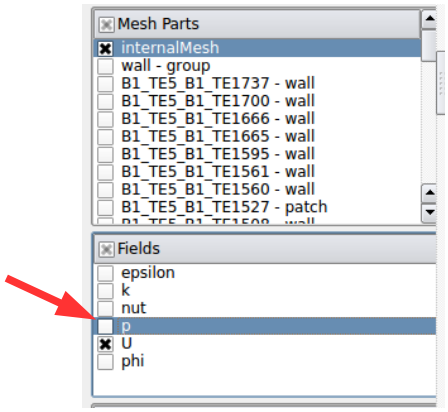
It should open this window:



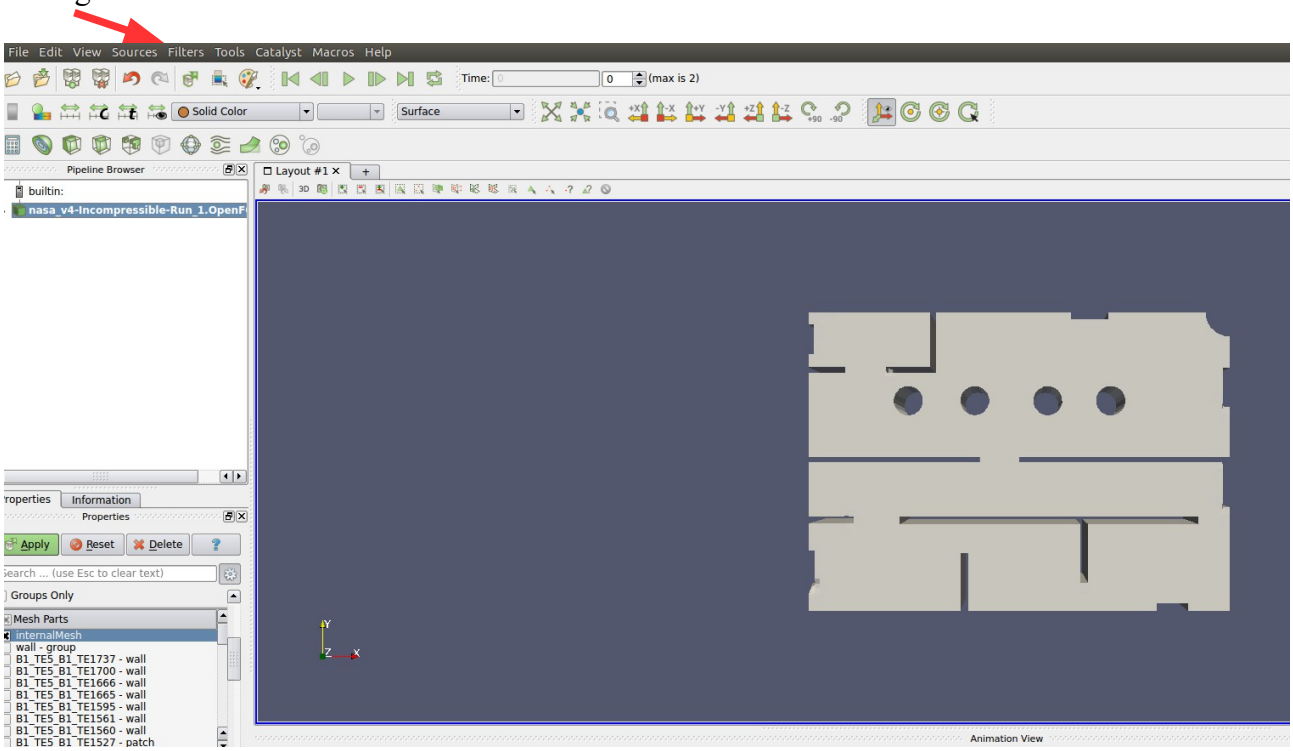
Roll the mouse to the Mesh Parts tab, remove the selections and let selected only internalMesh then **Apply** the mesh will be appear:



Remove the pressure field from the data, it is below the Mesh Parts window then **Apply** again.

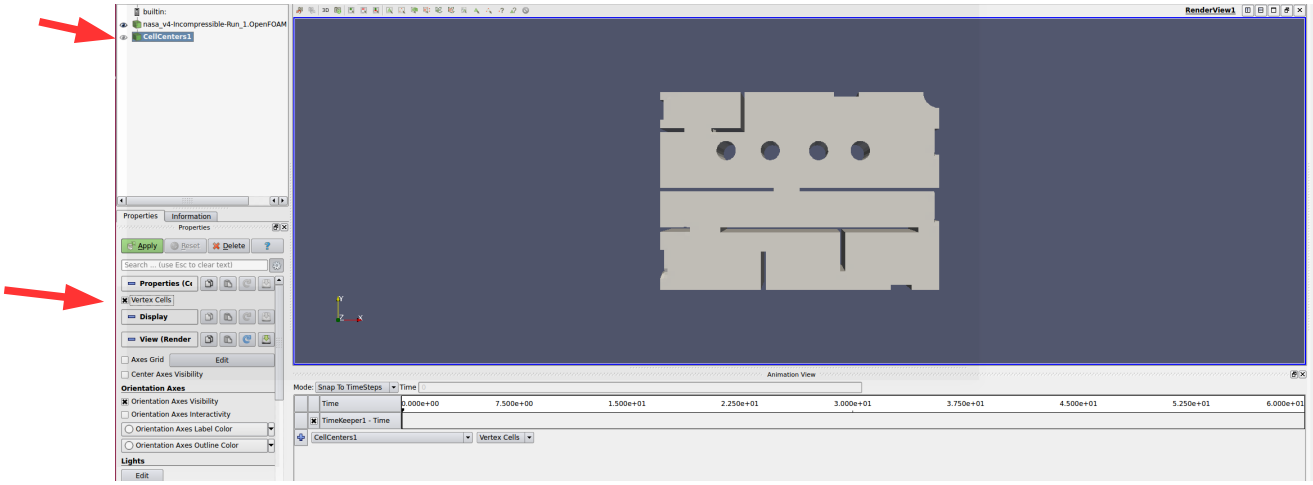


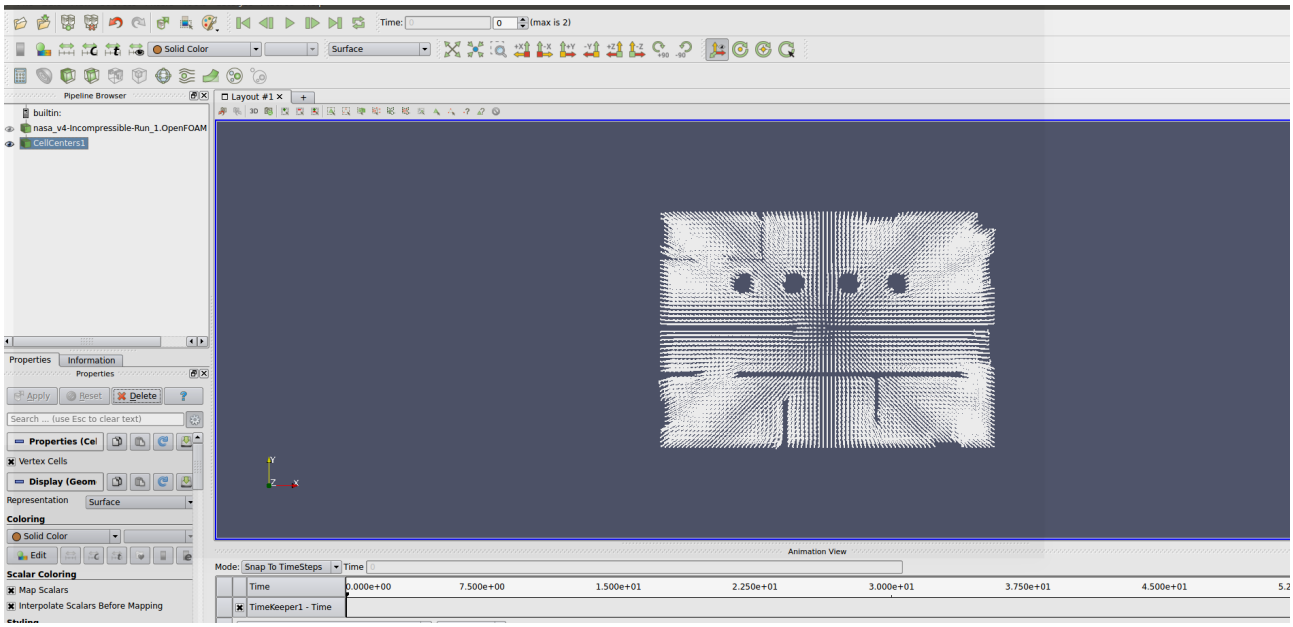
Now go to Filters tab.



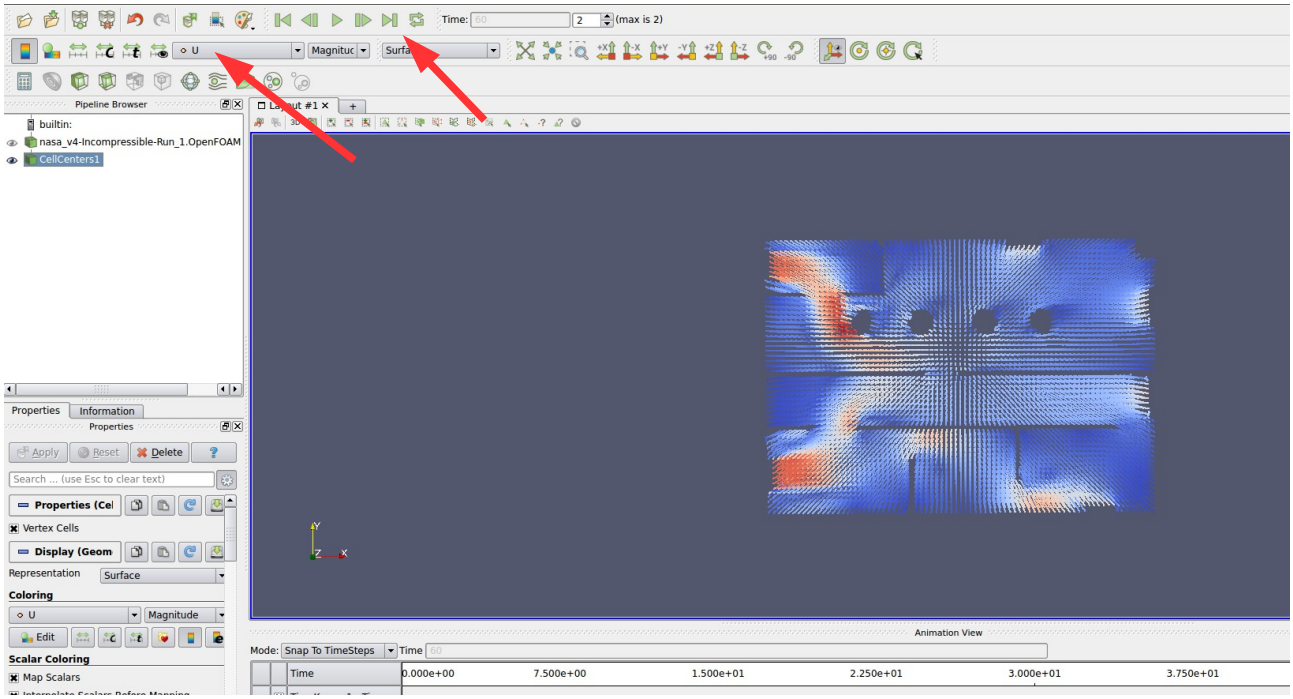
Filters → Alphabetical → Cell Centers

Should be appear CellCenters on Pipeline Browser, thenVextex Cells will be deselected we need to select after this **Apply**.

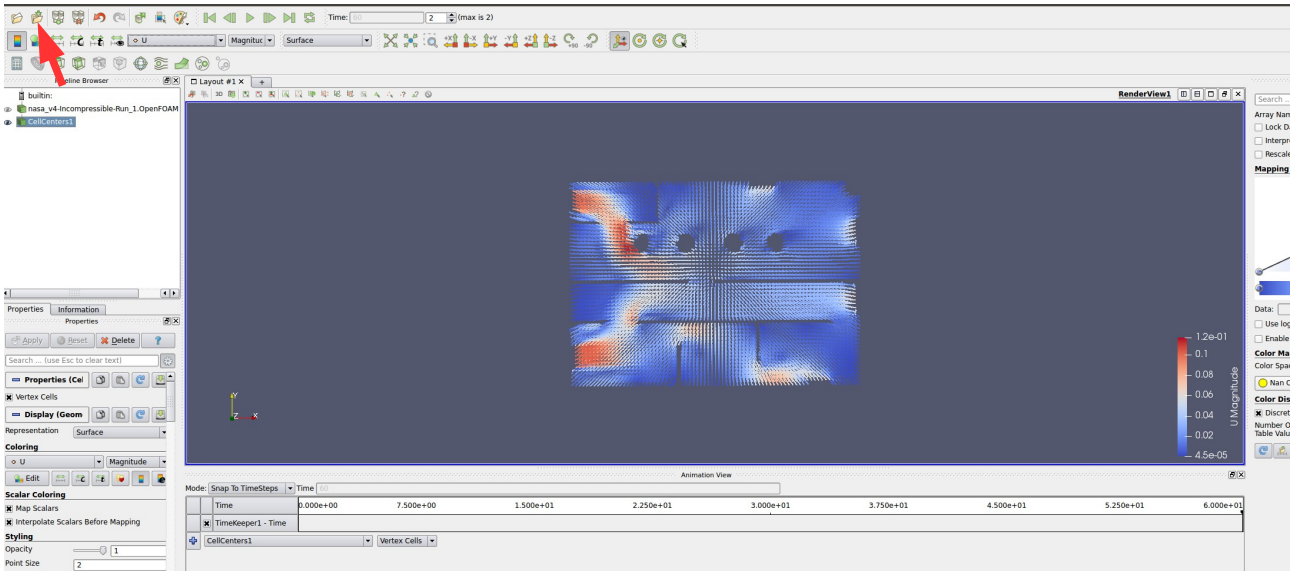




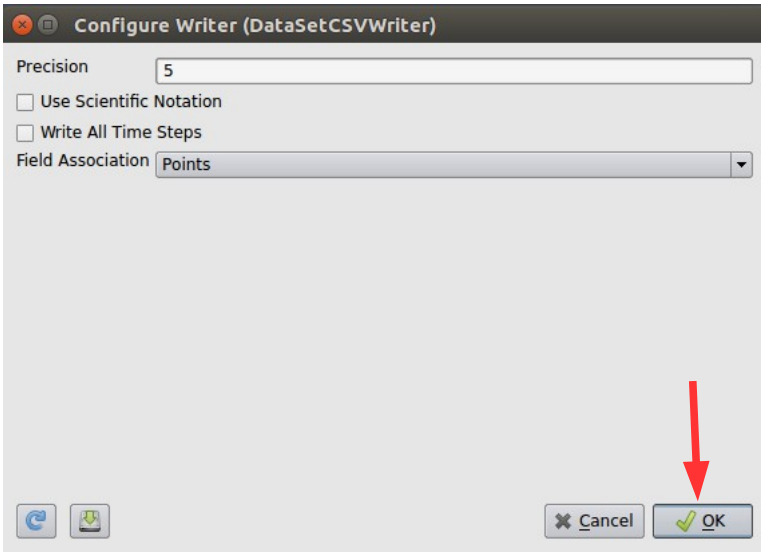
Change **Solid Color** to **U** and the time for the end like the image shows.



Save the data:



Select the name and don’t need to change the configurations



The file must contain U(0,1,2), Points(0,1,2)

```
"U:0","U:1","U:2","Points:0","Points:1","Points:2"
-0.00029794,-0.00037985,-1.591e-05,9.7089,3.6196,0.090111
-0.00036575,-0.00049408,-0.00012463,9.702,3.6305,0.26373
-0.00049584,-0.0027828,-6.2497e-05,9.7285,3.4669,0.075409
-0.00076093,-0.00028663,0.00013651,9.8333,3.6752,0.087251
-0.00026166,-0.00062103,1.504e-05,9.7027,3.6323,0.43333
-0.00054045,-0.0029571,-0.00010681,9.721,3.4775,0.2418
-0.00070536,-0.00044894,0.00010524,9.8144,3.6884,0.26125
-0.00075073,-0.0043384,-9.3076e-05,9.7373,3.3018,0.073091
-0.0015633,-0.0024667,4.2123e-05,9.8606,3.4946,0.066288
-0.0030601,-0.00060432,0.00013153,10.034,3.6818,0.080387
-0.00023921,-0.00087884,-7.4326e-05,9.703,3.6328,0.60529
-0.0005483,-0.0028279,-2.9297e-05,9.7235,3.4797,0.42279
-0.00064619,-0.00041714,-2.5167e-05,9.8096,3.6885,0.43427
-0.00079353,-0.0044018,-7.7616e-05,9.7339,3.3048,0.2369
-0.0015829,-0.0023594,-4.2042e-05,9.8462,3.512,0.23021
-0.002972,-0.00060926,-2.2595e-05,10.013,3.6944,0.25124
-0.00082341,-0.0063742,-9.0599e-05,9.7401,3.1316,0.072713
-0.0019519,-0.0042072,6.5834e-06,9.8727,3.3133,0.064168
-0.0032998,-0.0023246,8.2176e-05,10.036,3.5059,0.066422
-0.0046201,-0.00045529,8.5613e-05,10.21,3.6835,0.077722
-0.00032669,-0.0004573,1.8824e-05,9.703,3.6329,0.77614
-0.00049131,-0.0028233,-2.6992e-05,9.7244,3.4805,0.60057
-0.00068377,-0.00038861,-2.5235e-05,9.808,3.6883,0.60614
-0.00082818,-0.0043359,-1.6814e-05,9.7386,3.3061,0.41857
-0.0015464,-0.002327,-3.4609e-05,9.8458,3.5136,0.41965
-0.0028072,-0.00058258,-6.8114e-05,10.003,3.6941,0.4298
-0.00094017,-0.0063904,-7.6507e-05,9.7381,3.1324,0.23575
-0.0019939,-0.0041585,-6.5239e-05,9.8674,3.3189,0.224
-0.0031867,-0.0021786,-2.652e-05,10.024,3.5241,0.2293
-0.0044659,-0.00044402,-3.9904e-05,10.199,3.6955,0.24638
-0.00094545,-0.0086214,-0.00010152,9.7411,2.9608,0.072671
-0.0021963,-0.0063177,4.6708e-06,9.8772,3.1363,0.064103
-0.0037867,-0.0038845,3.4438e-05,10.04,3.3206,0.065917
-0.0047531,-0.0020028,2.5672e-05,10.211,3.5097,0.066125
```

Package Configuration

Create a new folder into gaden/test_env for this tutorial will be named tutorial. Open tutorial folder and create the folders:
- cad_models: Paste into cad_models our STL files.
- launch: copy launch files of MAPIRlab
- wind_simulations/W1: Paste the archive generated of OpenFoam (.csv). Rename this file to end **_0.csv**

Rename:
GADEN.launch to → tutorial.launch
GADEN_player.launch to → delete (unnecessary)
GADEN_preprocessing.launch → tutorial_preprocessing.launch
GADEN_simbot.launch → tutorial_simbot.launch

Launch Configurations
Replace the red words with green words

All launchs

Scenario Name:
line 6
<arg name="scenario" default="MAPIRlab" />
<arg name="scenario" default "tutorial" />

tutorial_preprocessing.launch

Configuration of scenario:
Obs: These files has to be in the formart ASCII not binary.
line 13
```
<!--#CAD models of the enviroment (.stl)-->
      <param name="number_of_models" value="3"/>
      <param name="model_0" value="$(find test_env)/$(arg scenario)/cad_models/MAPIRlab_walls.stl"/>
      <param name="model_1" value="$(find test_env)/$(arg scenario)/cad_models/MAPIRlab_tables.stl"/>
      <param name="model_2" value="$(find test_env)/$(arg scenario)/cad_models/MAPIRlab_wardrobes.stl"/>

<!--#CAD models of the enviroment (.stl)-->
      <param name="number_of_models" value="1"/>
      <param name="model_0" value="$(find test_env)/$(arg scenario)/cad_models/nasa_v4_ascii.stl"/>
```

line 19
```
<!--#CAD model of the outlets (.stl)-->
       <param name="number_of_outlet_models" value="2"/>
      <param name="outlets_model_1" value="$(find test_env)/$(arg scenario)/cad_models/MAPIRlab_doors.stl"/>
      <param name="outlets_model_0" value="$(find test_env)/$(arg scenario)/cad_models/MAPIRlab_windows.stl"/>

<!--#CAD model of the outlets (.stl)-->
      <param name="number_of_outlet_models" value="0"/>
```

Set wich a empty point to the simulation now what local its air.
line 24

```
<!--#3D Location of a point in free-space-->
      <param name="empty_point_x" value="1.0"/>
     <param name="empty_point_y" value="1.0"/>
     <param name="empty_point_z" value="0.5"/>

<!--#3D Location of a point in free-space-->
      <param name="empty_point_x" value="7.0"/>    <!-- ### (m)-->
     <param name="empty_point_y" value="5.0"/>    <!-- ### (m)-->
     <param name="empty_point_z" value="0.5"/>
```

Wind File

line 30

```
<!--#Wind Data (the node will append _i.csv to the name that is specified here)-->
    <param name="wind_files" value="$(find test_env)/$(arg scenario)/wind_simulations/W1/wind_at_cell_centers"/>

<!--#Wind Data (the node will append _i.csv to the name that is specified here)-->
    <param name="wind_files" value="$(find test_env)/$(arg scenario)/wind_simulations/W1/nasa_v4"/>
```

Roslaunch:

$ roslaunch test_env tutorial_preprocessing.launch

If all goes well, the following will appear:

```
process[rosout-1]: started with pid [4864]
started core service [/rosout]
process[preprocessing-2]: started with pid [4867]
Filling...
[ INFO] [1628538806.170963625]: Preprocessing done
```

After INFO Preprocessing done… Press **Ctrl+c** and check:

tutorial/occupancy.pgm
tutorial/occupancy.yaml
tutorial/OccupancyGrid3D.csv
tutorial/wind_simulations/W1/nasa_v4_0.csv_U
tutorial/wind_simulations/W1/nasa_v4_0.csv_V
tutorial/wind_simulations/W1/nasa_v4_0.csv_W

tutorial.launch

This launch file generate a simulation/dispersion files. Creating a “tutorial/gas_simulations/W1/ FilamentSimulation_gasType_0_sourcePosition_ **source_location_ x_source_location_y_ source_location_z**” files.

Setting the source location. It must be a coordinate inside the map

line 8

```
  <arg name="source_location_x" default="2.50" />
```

```
  <arg name="source_location_y" default="-3.50" />
  <arg name="source_location_z" default="1.00" />

   <arg name="source_location_x" default="2.00" />
  <arg name="source_location_y" default="2.00" />
  <arg name="source_location_z" default="1.00" />
```

Importing files to visual simulation.
OBS: Unlike the previous case where the file had to be ASCII, now it must be binary or it won't work all of simulation.

line 18

```
<!--# Plot CAD models (Collada format .dae)-->
            <param name="number_of_CAD" value="3"/>
            <rosparam subst_value="True">
        CAD_0: package://test_env/$(arg scenario)/cad_models/10x6_walls.dae
        CAD_0_color: [0.92, 0.96, 0.96]

        CAD_1: package://test_env/$(arg scenario)/cad_models/10x6_door_left.dae
        CAD_1_color: [0.96, 0.17, 0.3]

        CAD_2: package://test_env/$(arg scenario)/cad_models/10x6_door_right.dae
        CAD_2_color: [0.96, 0.17, 0.3]

            CAD_3: package://test_env/$(arg scenario)/cad_models/MAPIRlab_tables.stl
        CAD_3_color: [0.92, 0.96, 0.96]
      </rosparam>

 <!--# Plot CAD models (Collada format .dae)-->
            <param name="number_of_CAD" value="1"/>
            <rosparam subst_value="True">
        CAD_0: package://test_env/$(arg scenario)/cad_models/nasa_v4.stl
        CAD_0_color: [0.92, 0.96, 0.96]
```

Time of simulation

line 53

```
 <param name="sim_time" value="300" />              <!--### [sec] Total time of the gas dispersion simulation--
>

<param name="sim_time" value="500" />              <!--### [sec] Total time of the gas dispersion simulation-->
```

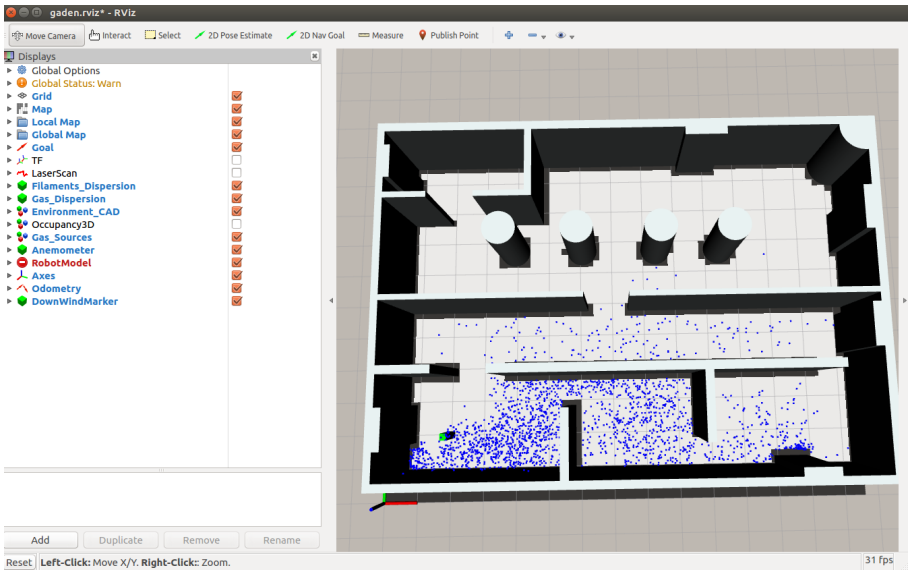
line 69

```
<!--# WindFlow data (from CFD)-->
            <param name="wind_data" value="$(find test_env)/$(arg scenario)/wind_simulations/$(arg
simulation)/wind_at_cell_centers_" />
            <param name="wind_time_step" value="1" />          <!--### (sec) time increment between Wind
snapshots-->

<!--# WindFlow data (from CFD)-->
            <param name="wind_data" value="$(find test_env)/$(arg scenario)/wind_simulations/$(arg
simulation)/nasa_v4_" />
            <param name="wind_time_step" value="1" />          <!--### (sec) time increment between Wind
snapshots-→
```

Roslaunch

$ roslaunch test_env tutorial.launch



Wait until the filaments of the simulation freeze, which is the moment when the code finish generating the files. After that **Ctrl+c**.

If everything ok will be generated.

tutorial/gas_simulations/W1/FilamentSimulation_gasType_0_sourcePosition_2.00_2.00_1.00/ … from iteration_0 to iteration_500 (in our case)

tutorial/gas_simulations/W1/FilamentSimulation_gasType_0_sourcePosition_2.00_2.00_1.00/wind/ wind_iteration_0

tutorial_simbot.lauinch

Source position, this part search the folder named FilamentSimulation_gasType_0_sourcePosition_ **source_location_x_source_location_y_ source_location_z** if it be wrong the simulation don't will be work.
line 12

```
<arg name="source_location_x" default="2.50" />
<arg name="source_location_y" default="-3.50" />
<arg name="source_location_z" default="1.00" />

<arg name="source_location_x" default="2.00" />
<arg name="source_location_y" default="2.00" />
<arg name="source_location_z" default="1.00" />
```

Importing files to visual simulation.
OBS: Unlike the previous case where the file had to be ASCII, now it must be binary or it won't work all of simulation.

line 56

```
<!--# Plot CAD models (Collada format .dae)-->
      <param name="number_of_CAD" value="3"/>
      <rosparam subst_value="True">
         CAD_0: package://test_env/$(arg scenario)/cad_models/MAPIRlab_walls.dae
         CAD_0_color: [0.32, 0.36, 0.36]
         CAD_1: package://test_env/$(arg scenario)/cad_models/MAPIRlab_doors.dae
         CAD_1_color: [1, 0, 0]
         CAD_2: package://test_env/$(arg scenario)/cad_models/MAPIRlab_windows.dae
         CAD_2_color: [1, 0, 0]
      </rosparam>

<!--# Plot CAD models (Collada format .dae)-->
      <param name="number_of_CAD" value="1"/>
      <rosparam subst_value="True">
         CAD_0: package://test_env/$(arg scenario)/cad_models/nasa_v4.stl
         CAD_0_color: [0.92, 0.96, 0.96]
      </rosparam>
```

Loop configuration

line 95

```
<param name="loop_from_iteration" value="15" />
<param name="loop_to_iteration" value="24" />

<param name="loop_from_iteration" value="150" />
<param name="loop_to_iteration" value="499" />
```

At the end of the code add

```
  <node pkg="tf" type="static_transform_publisher" name="od_broadcaster" args="0.0 0.0 0.0 0 0 0 map odom
100" />
```

Atention of:
Tutorial/launch/ros/stage.world

Map configurations:

```
line 128

floorMap                # load an environment bitmap
(
  name "SimulatedMap"

  bitmap "../../occupancy.pgm"
  size [14.5 10.8 3.00]          #m
  pose [7.25 5.4 0.000 0.000 ]    #Coordinates (m) of the Center of the image_map
)
```

Robot Initial Position

```
line 137

# throw in a robot
robotBase( pose [ 13.0 8.5 0.000 0.000 ] name "SimRobot" color "blue")
```

Roslaunch

$ roslaunch test_env tutorial_simbot.launch