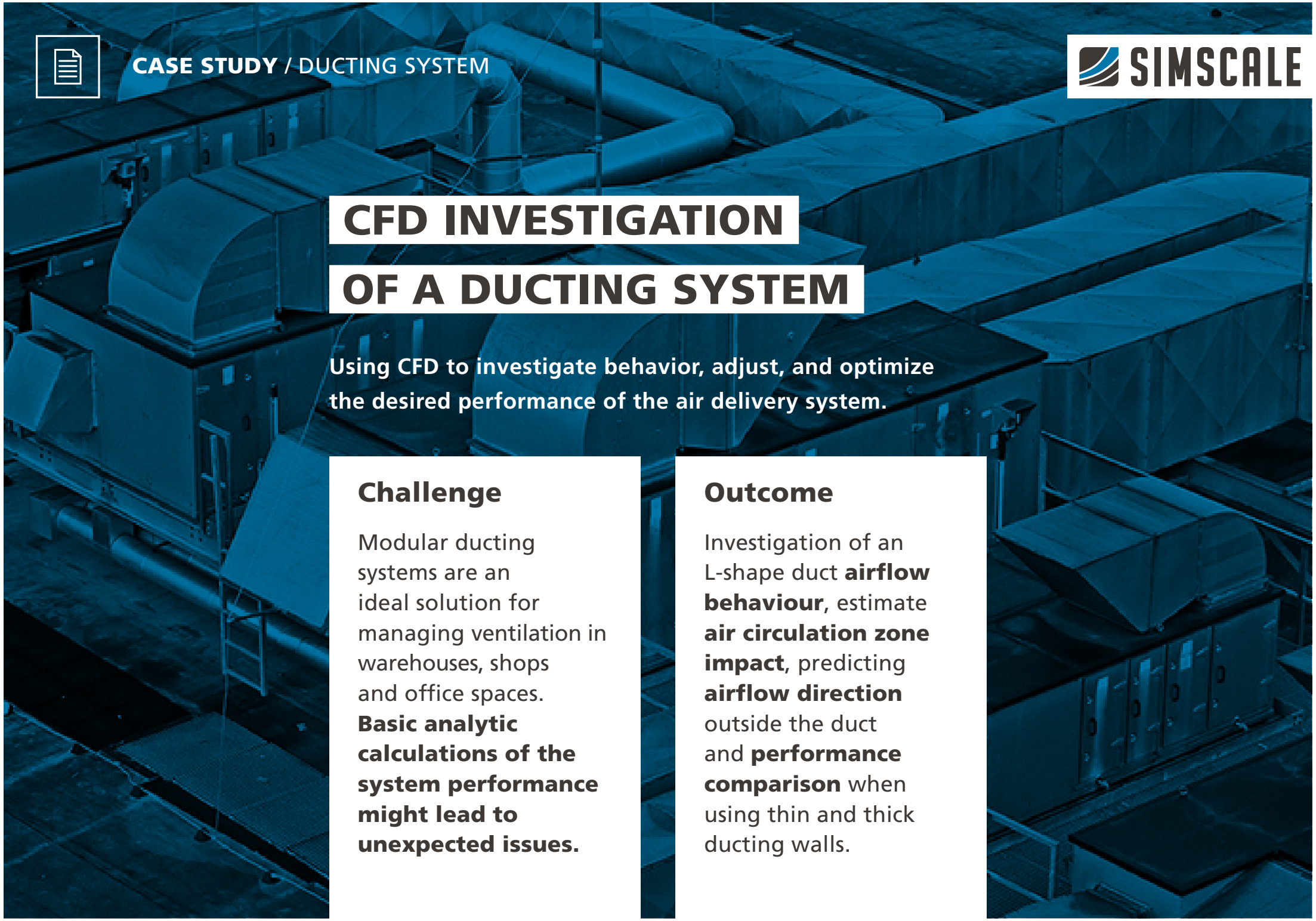


CASE STUDY / DUCTING SYSTEM

SIMSCALE

CFD INVESTIGATION OF A DUCTING SYSTEM

Using CFD to investigate behavior, adjust, and optimize the desired performance of the air delivery system.

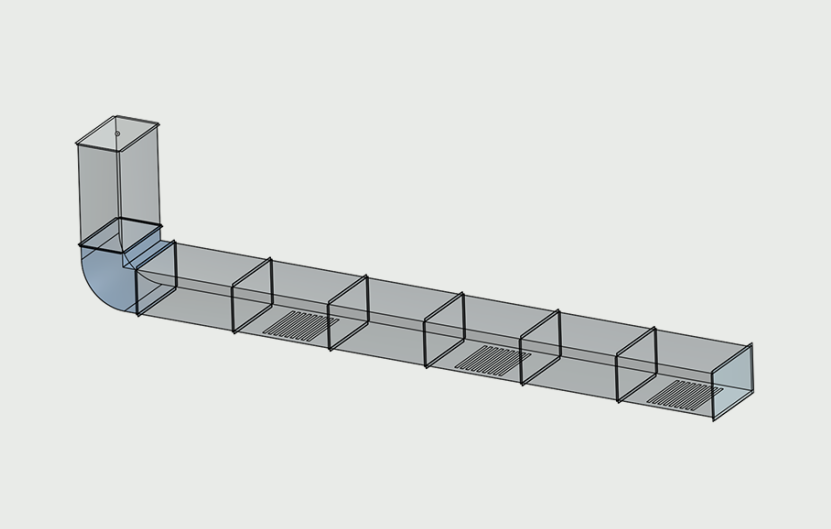
Challenge

Modular ducting systems are an ideal solution for managing ventilation in warehouses, shops and office spaces. **Basic analytic calculations of the system performance might lead to unexpected issues.**

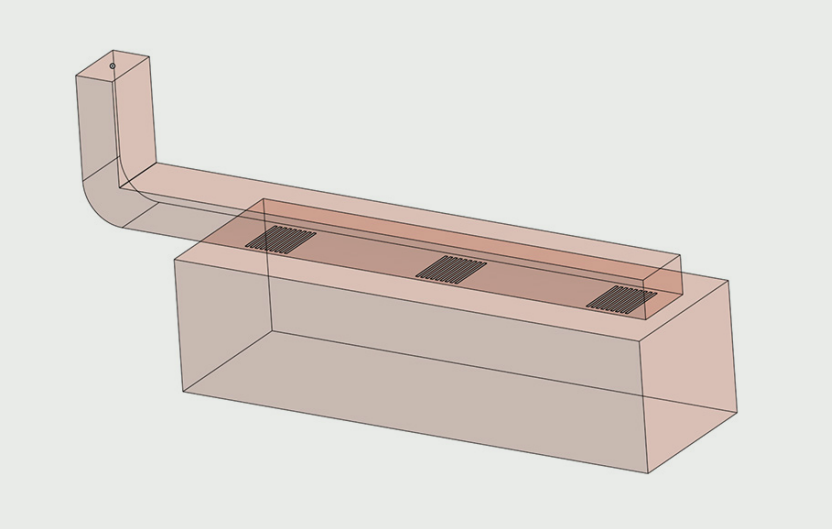
Outcome

Investigation of an L-shape duct **airflow behaviour**, estimate **air circulation zone impact**, predicting **airflow direction** outside the duct and **performance comparison** when using thin and thick ducting walls.

GEOMETRIES

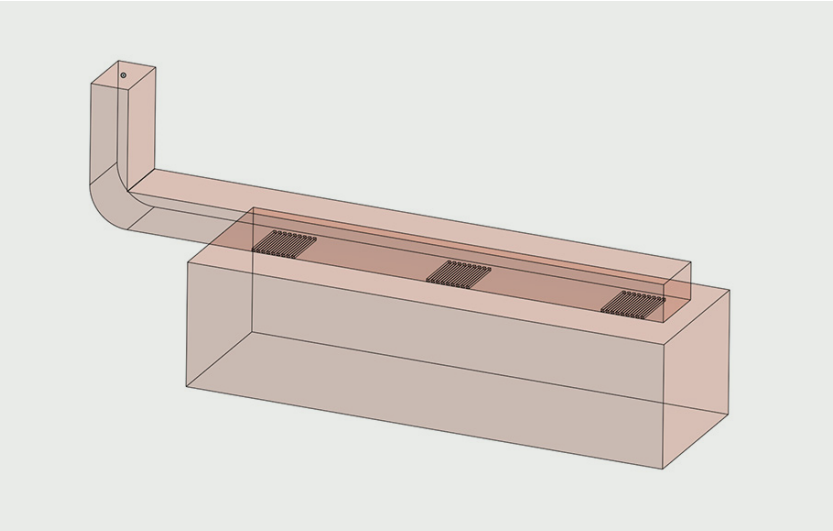


CAD Representation
of the Modular Ducting System.



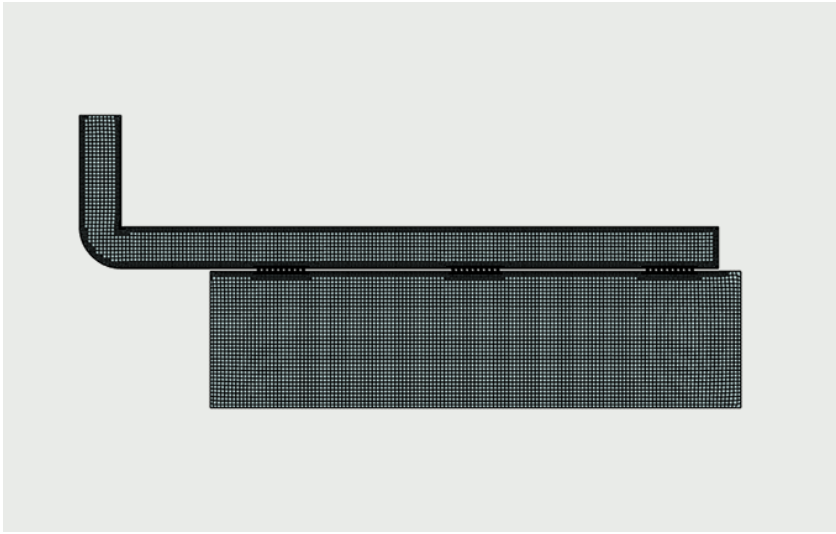
The CAD geometry needs to be transformed so that it can be used in a **fluid flow simulation**. The task involves making a volume extract of the air inside the duct and then adding extra area below the outlets. This could be thought as **“taking a negative” of the 3D design** and removing insignificant elements of the design.

GEOMETRIES

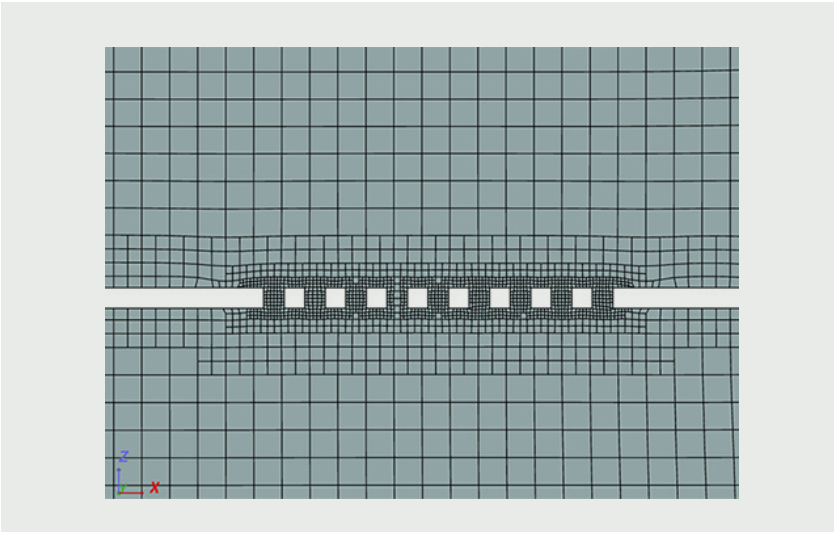


As an extra step in the geometry preparation, **an additional model is created**, in which the length of the outlets connecting the duct with the air below is extended. This would mimic a situation in which the **duct is covered with an extra insulation layer**.

MESHES

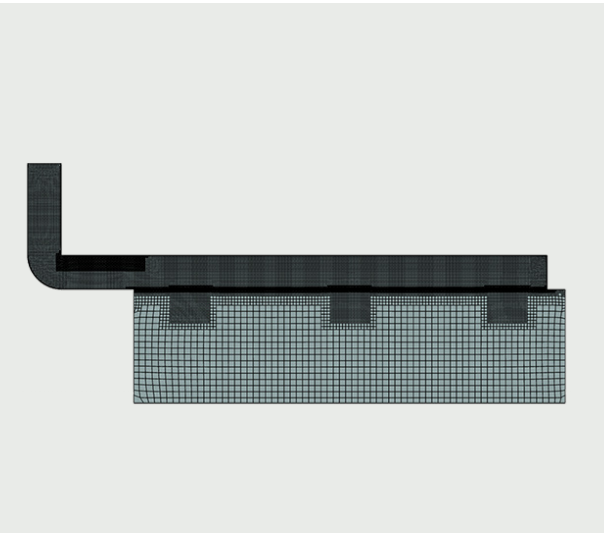


In order to facilitate an optimal simulation workflow, multiple meshes were created and used in parallel during the analysis. To speed up the meshing and analysis process, the symmetry of the domain was taken into account, allowing to reduce the domain size in half.



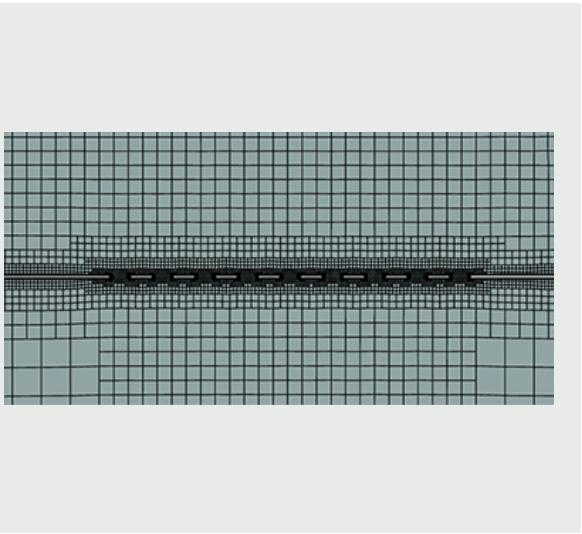
The very first, **coarse mesh consisted of only 430k elements**. It would be used for fast, crude simulation of the system aimed in locating zones of interest, such as circulation areas, back-flows, and jets. Note that the mesh resolution is not sufficient to allow drawing decisive conclusions from the simulations performed on it. A properly resolved mesh needs to be prepared.

MESHES

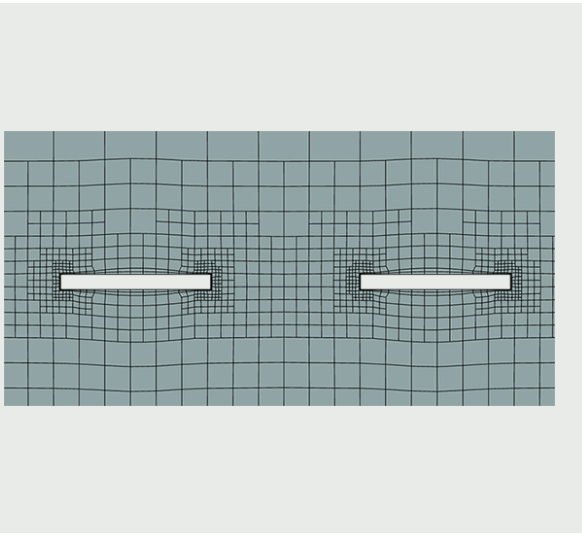


Thin-Wall Domain, 5.25 mln Elements

With the coarse mesh ready, the ones that will be used for a complete analysis can be prepared.

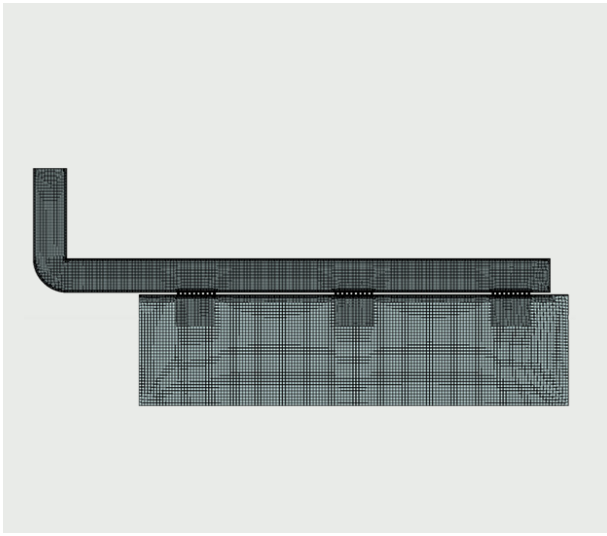


Additional refinements in the expected recirculation zone after the 90-degree bend were added, together with an overall refinement of the flow within the



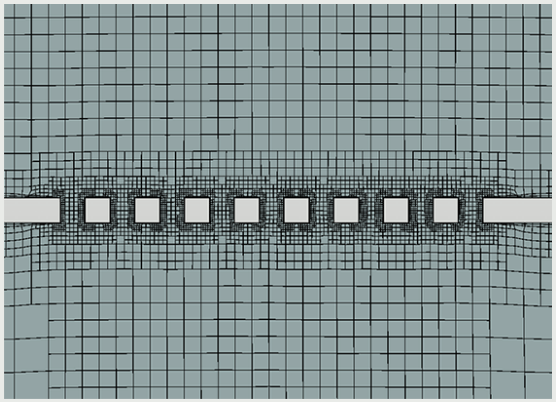
duct, further refinements of the outlet sections, and finally, mesh refinement in the expected jet areas.

MESHES

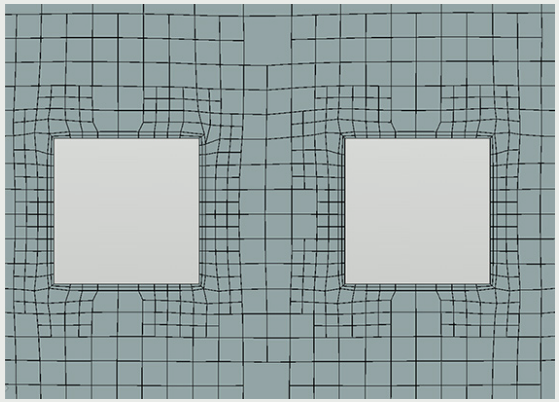


Thick Wall Domain, 1.95 mln Elements

With the coarse mesh ready, the ones that will be used for a complete analysis can be prepared.

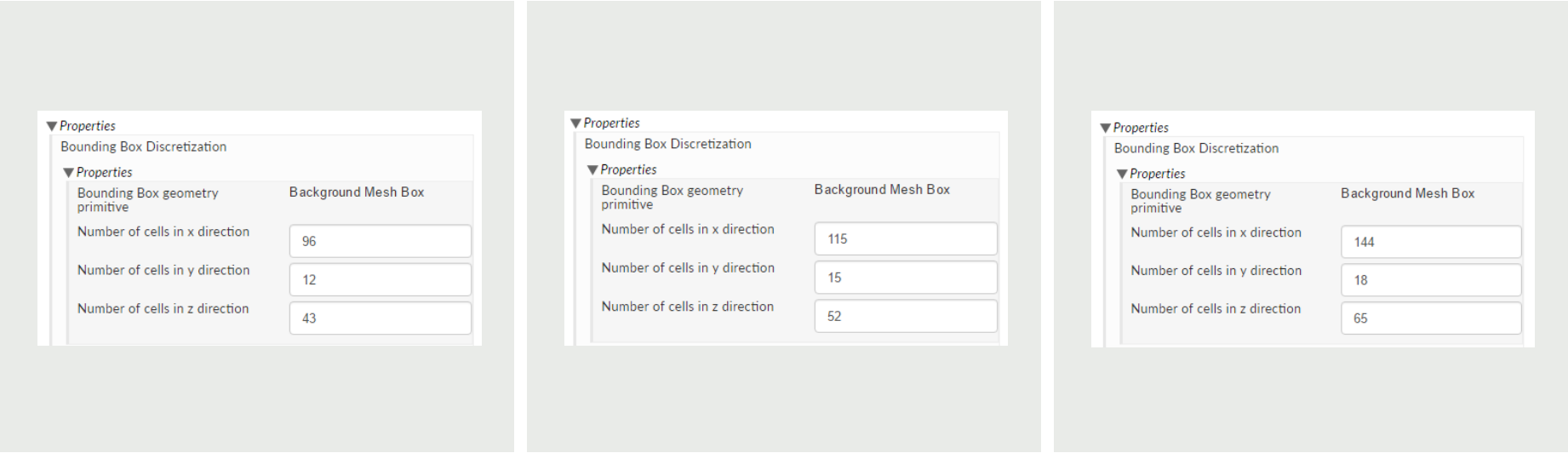


Additional refinements in the expected recirculation zone after the 90-degree bend were added, together with an overall



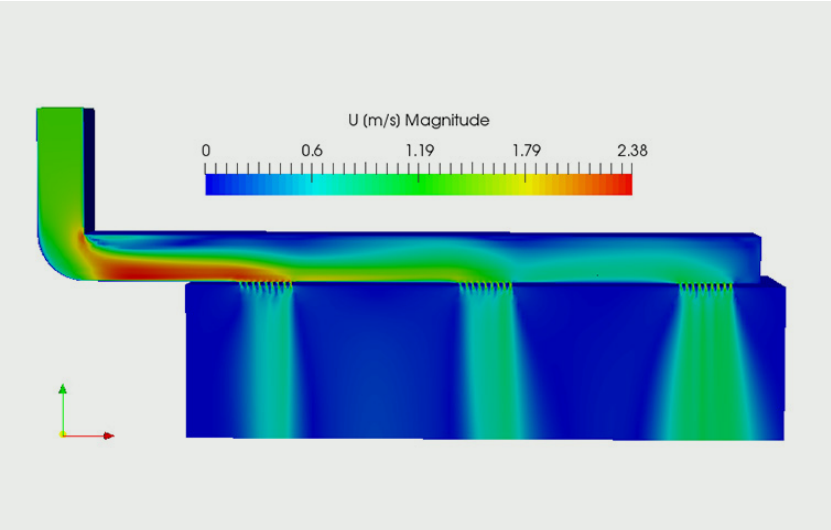
refinement of the flow within the duct, further refinements of the outlet sections, and finally, mesh refinement in the expected jet areas.

MESHES



Finally, to ensure that the obtained results are within the expected accuracy range, a **mesh independence study** was performed on the thin-wall geometry, which involved increasing the number of elements in XYZ direction by 20% and by 50%.

SIMULATION



The simulations were carried on SimScale using **incompressible steady-state analysis** with the k-omega SST turbulence model.

The initial results obtained from the quick 400k volume mesh allowed to easily identify the zone where expected recirculation after 90-degree bend would appear.

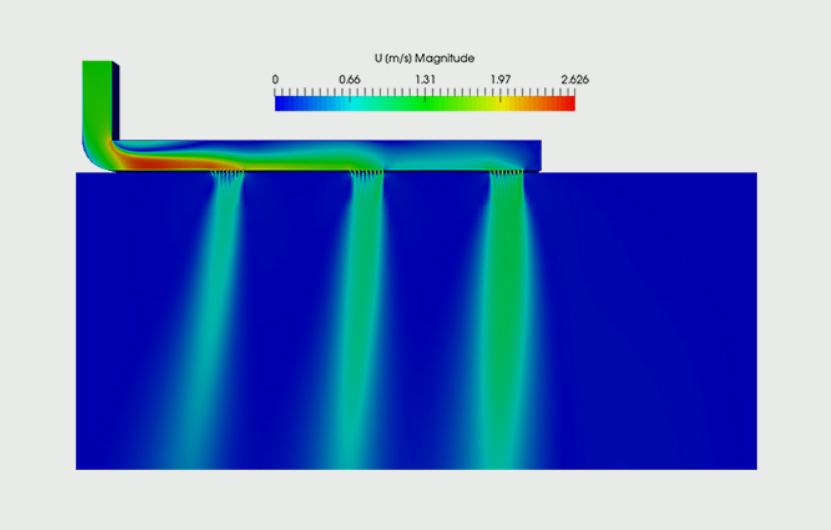
FLOW MEDIUM	**INLET***	**AIR DOMAIN**
Standard conditions	0.2 m3/s through the duct	The bottom and sidewalls were set as open outlets with 1atm of total pressure

* this meant using 0.1 m3/s at inlet condition since symmetry was applied

The initial simulation on the coarsest mesh was carried on 8 core machine and took 14 minutes to complete. Full-scale simulations were carried on 32 core machines and took:

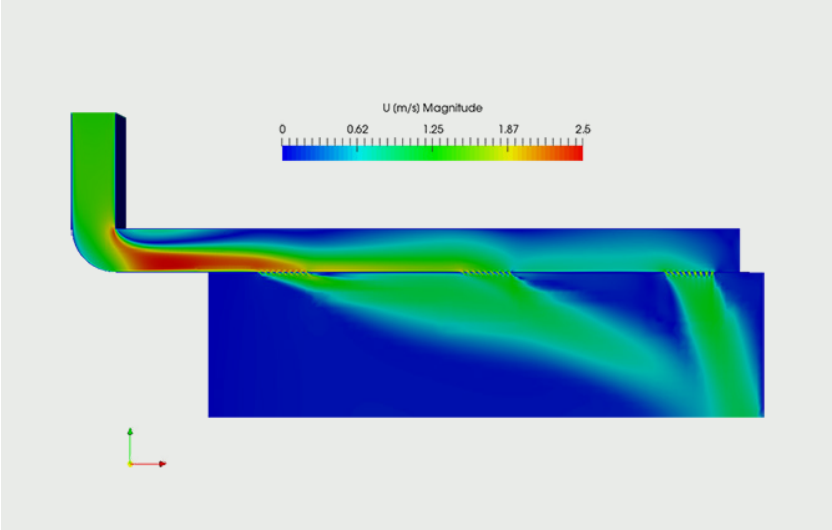
- 60 minutes for the thick wall domain with a mesh of 1.9 mln elements
- 114 minutes, for thin wall domain with a mesh of 5.25 mln volumes
- 174 minutes for 20% increased base mesh size
- 432 minutes for 50% increased base mesh size

RESULTS



Performing the analysis quickly, mesh refinements were prepared in order to properly resolve the flow within the duct.

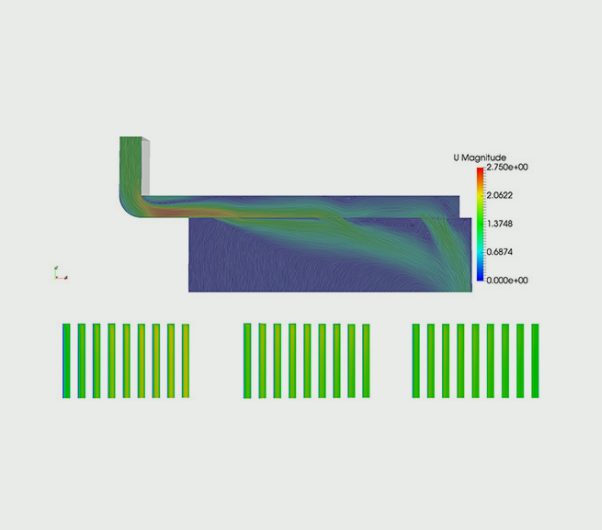
The **thick wall ducting system** followed the behavior predicted by the first calculation and provided insights to the flow strength coming out of the ducts.

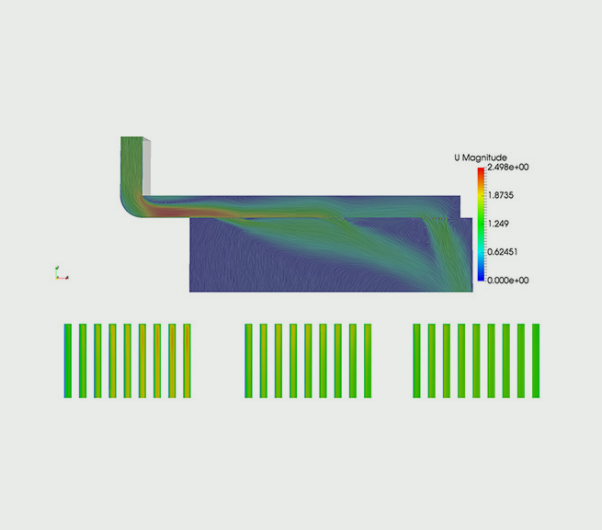


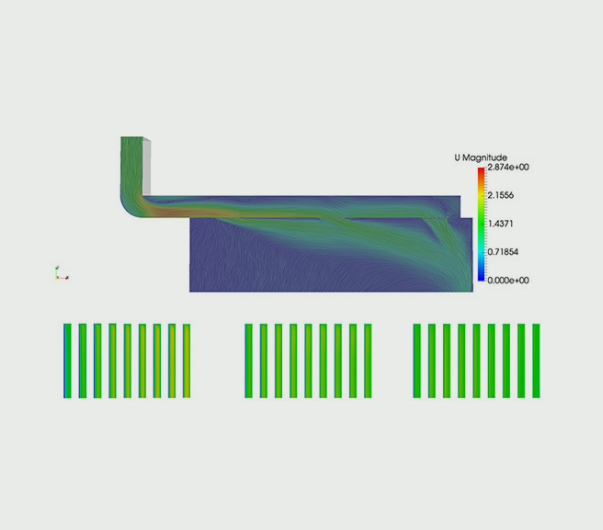
Interestingly, the flow patterns coming from the simulations of the **thin-wall duct** have shown completely different patterns.

The results for both duct configurations show the disproportion of the flow rate exiting through each of the 3 shatters. It clearly indicates that there is more flow exiting through the last shatter and the least through the first one.

RESULTS



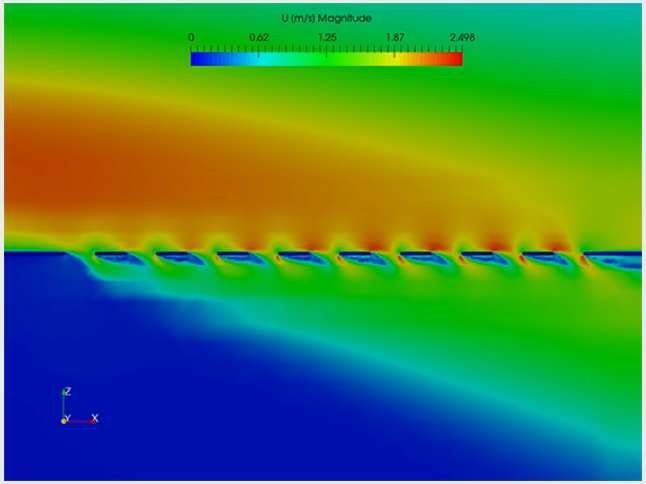




A further investigation of the **thin-wall duct** during the **mesh independence study** have revealed that the obtained flow field is valid and that another explanation of the difference is required.

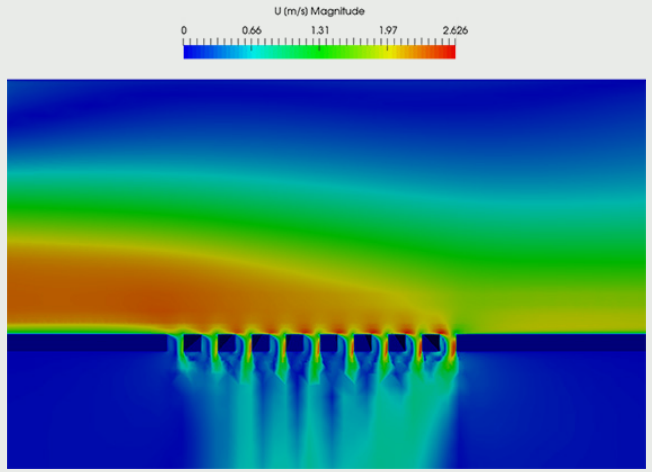
RESULTS

thin wall



A detailed investigation of the flow near the "shatter" outlets has shown that the **thickness of the duct wall** has a significant impact on the outgoing jet direction. The first clue comes from the fast velocity area created after 90-degree bend. This area is created due to the presence of flow separation and redirection, as well as being pushed down by the recirculation vortex above it.

thick wall



This jet of fast-moving air arrives to the first shatters and tries to maintain its horizontal motion. In the **thin wall duct case**, the shutters "slice" the flow and allow it to maintain part of its horizontal momentum. This results in the first jet to exit the duct at a significant angle. On the other side, the **thick wall duct** prevents the "slicing" effect, since the shutter walls are long enough to completely redirect the flow downwards.

CONCLUSION

The comparison of "shutter" outlet areas proves that the **thickness of the duct wall as well as the outlet area position have a significant impact on the desired performance** of the air delivery system.

Find this project and use it as a template here.