Cooling studies for FCC-ee vertex detector

MAPS detectors technologies for the FCC-ee vertex detector

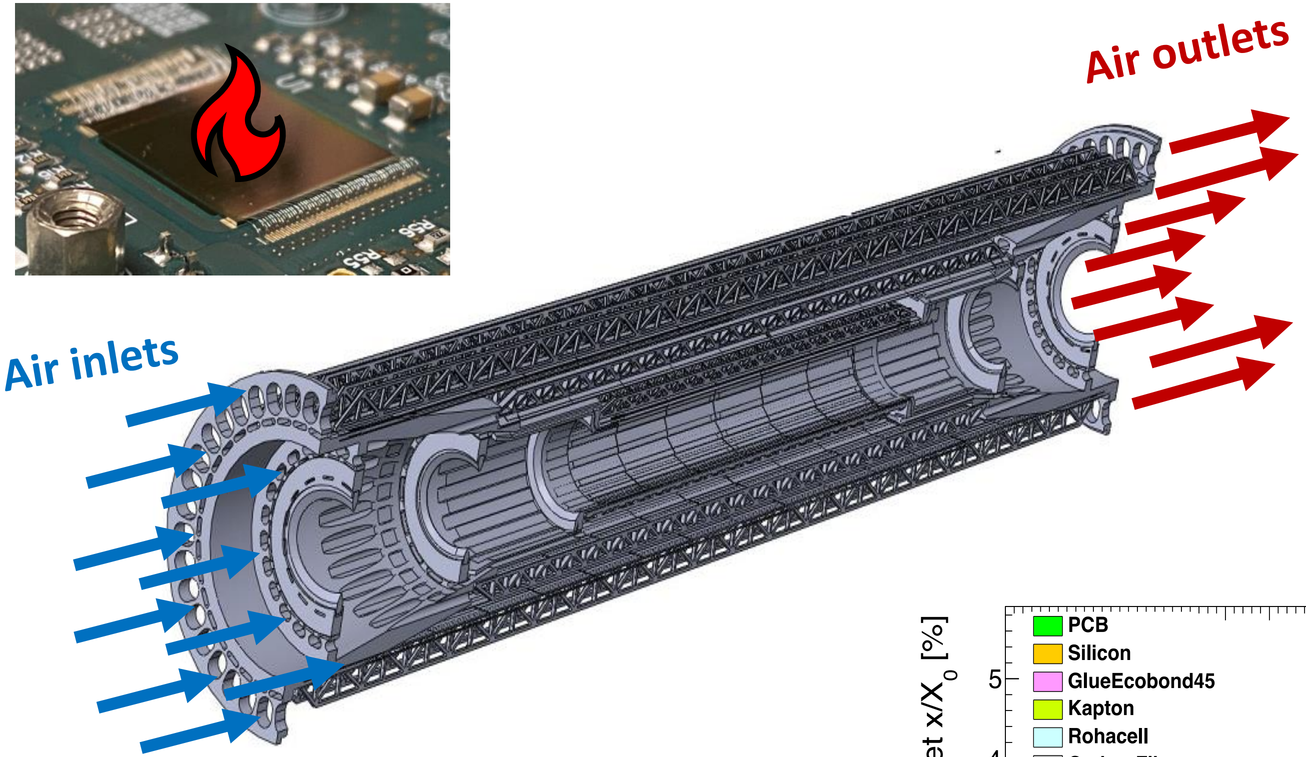
CERN, 1 - 2 July 2024

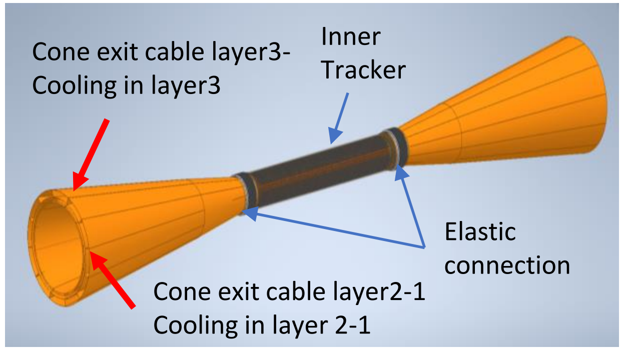
Giorgio Baldinelli (1) (2), Filippo Bosi (3), Fabrizio Palla (3), Giulia Pascoletti (1) (2), Cristiano Turrioni (2)

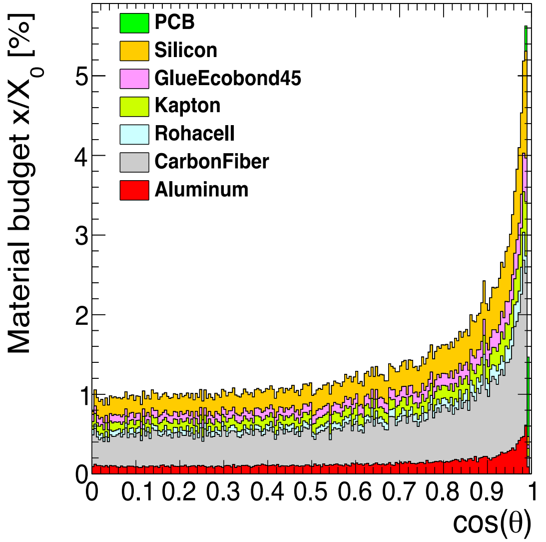
(1) *Università degli Studi di Perugia*
(2) *Istituto Nazionale di Fisica Nucleare – INFN - Sezione di Perugia*
(3) *Istituto Nazionale di Fisica Nucleare – INFN - Sezione di Pisa*

AN AIR COOLING SYSTEM FOR THE VERTEX DETECTOR

- A cooling system is necessary for removing the heat dissipated by sensors and electronics inside the vertex detector volume.
- A system based on the forced convection of a gas inside the detector seems particularly promising.
- This would substantially reduce material budget.
  - No pipes needed inside the vertex volume, just the gas flow.
  - Services concentrated in the two endcaps.
- Air is the easiest fluid to use on this purpose.







HOW TO EVALUATE THE THERMAL PERFORMANCE ?

- Two different ways to proceed:

NUMERICAL SIMULATIONS

The first approach

EXPERIMENTAL TEST

- ➢ Faster to have results.
- ➢ Easier to compare different design choices.
- ➢ Generally less expensive.

- Plenty of software are available, both Licensed and Open Source.

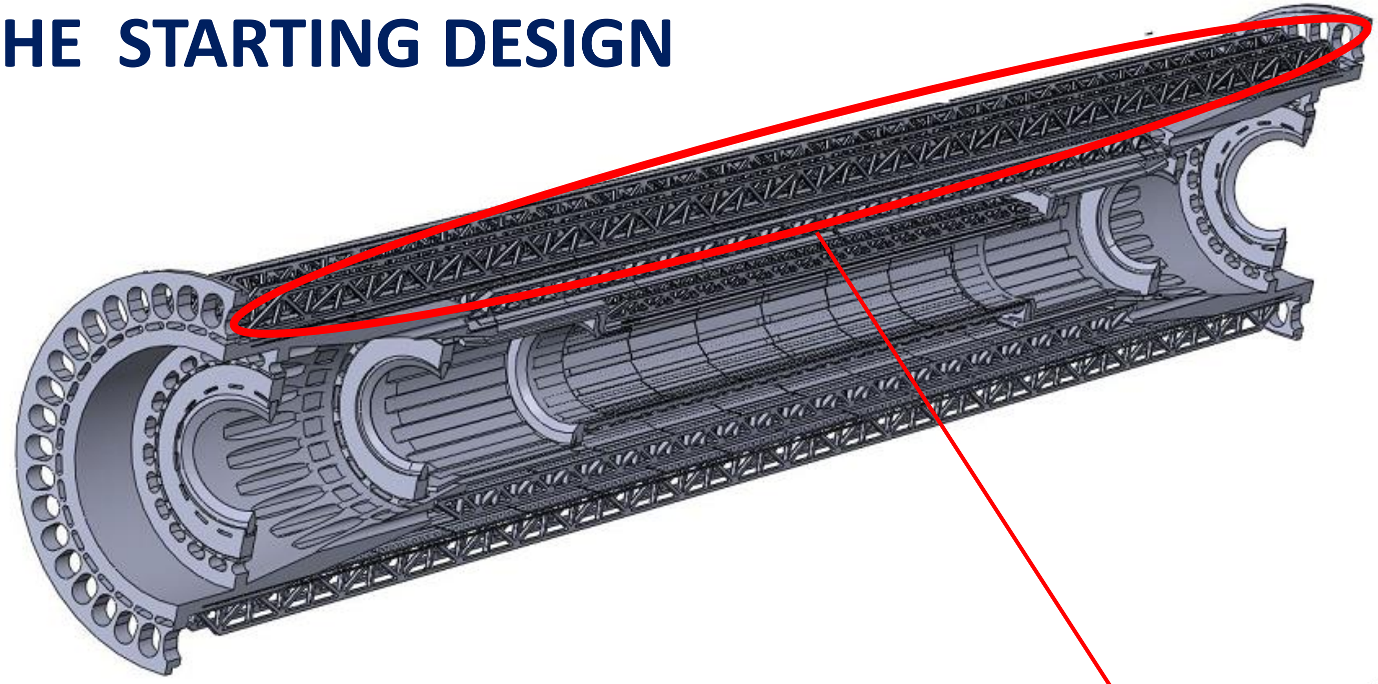
Ansys

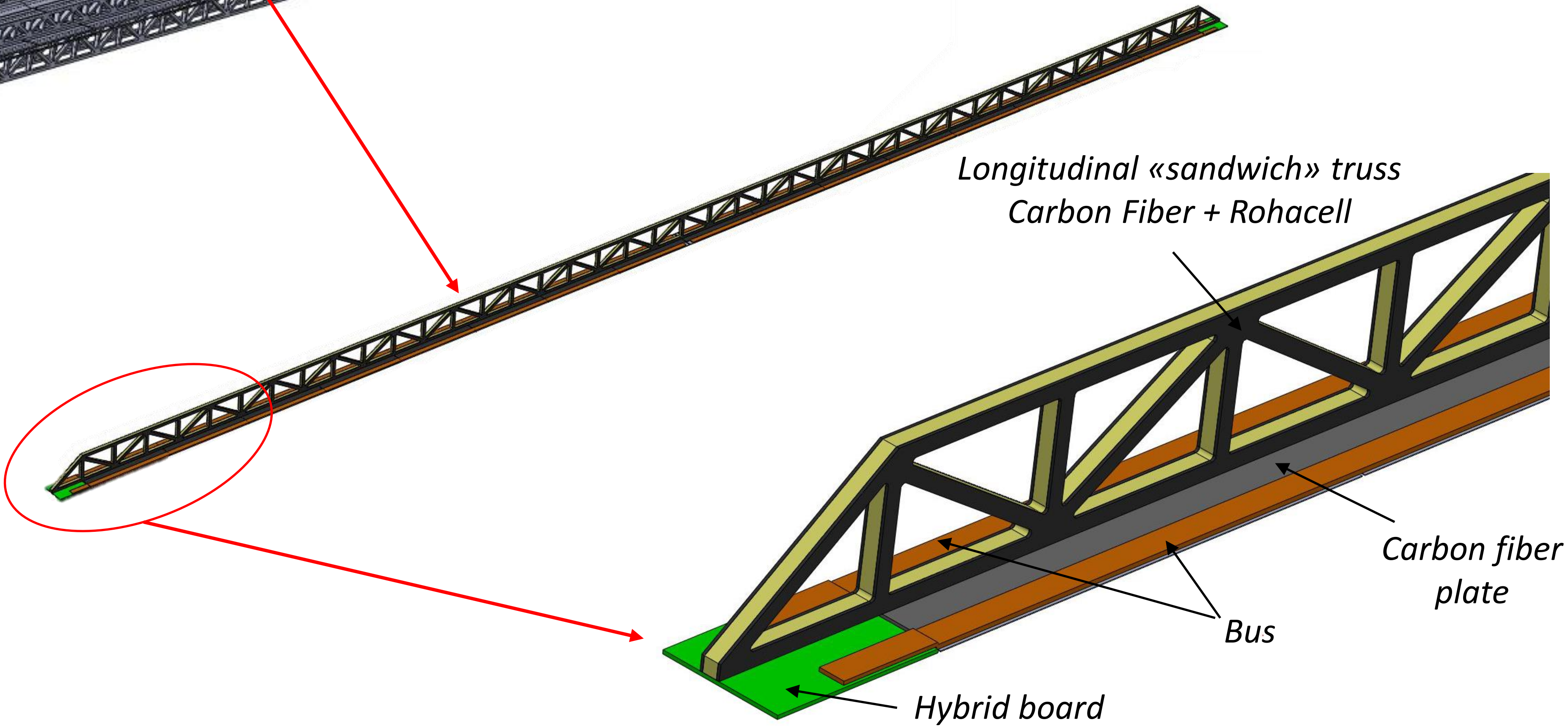
- ➢ We used the Ansys suite, mainly because of the greater experience of our team working with it.

THE STARTING DESIGN

- The detector is made of lightweight longitudinal elements ("staves") supported by endcaps at the ends:

  - 36 staves in layer 3, 16 Modules each, L = 540mm
  - 24 staves in layer 2, 10 Modules each, L = 340mm
  - 15 staves in layer 1, 6 Modules each, L = 220mm

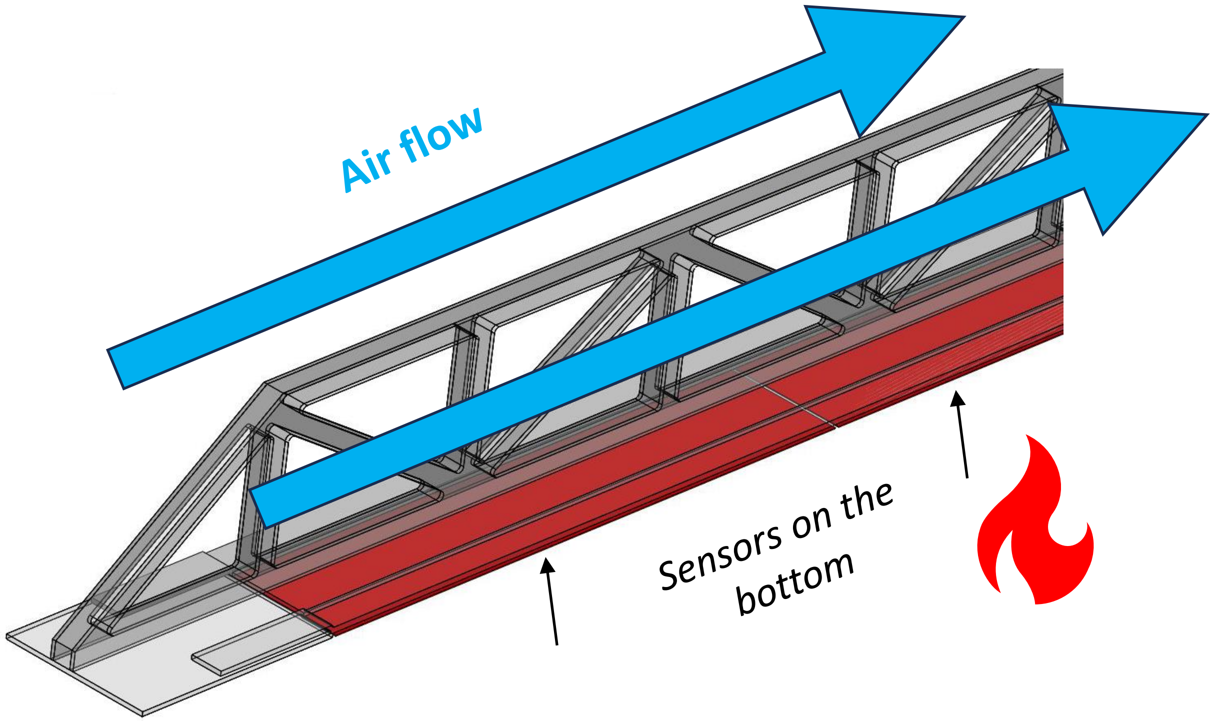




- Reticular lightweight support to provide stiffness.
- Thin carbon fiber walls interleaved with Rohacell.
- 2 buses (data and power).
- 1.8 mm wide and 250 µm thick (50 µm Al, 200 µm kapton) per side.
- Inspired to low mass hybrid R&D.
- Sensors facing interaction point w/o any other material in front.
- Readout chips either sides.

INPUTS FOR THE MODEL

- Sensors are on the bottom face of each stave.
- Current estimation for sensors power dissipation:
  - Layer 3: $\dot{Q}$ **~ 77 W** (total)
  - Layer 2: $\dot{Q}$ **~ 32 W** (total)
  - Layer 1: $\dot{Q}$ **~ 12 W** (total)



- Modeled as constant volumetric heat source inside the volume of sensors.
- For the moment, only the bulk of the electronics boards is represented.
- Air flow has the same direction for all the layers (co-current flow)
- Input for simulations:
  - Temperature of air at inlet: **T_{air} = 15°C**
  - Velocity of air at inlet: we tried different scenarios, with respectively: **v_{in_1}=10 m/s,** **v_{in_2}=20 m/s,** **v_{in_3}=30 m/s**

NUMERICAL ANALYSIS OF THE VERTEX

- ***Some points to have in mind:***

  - Different heat exchange modes are present :

    - Conduction - through the mechanical parts
    - Convection - with the coolant, by definition
    - Radiation - Probably negligible, but worth evaluating

→ ***«Conjugate heat transfer problem»***
Need to include in the model both the solid and the fluid domain, and make a mesh for both.

- Large difference in scale between the various components.

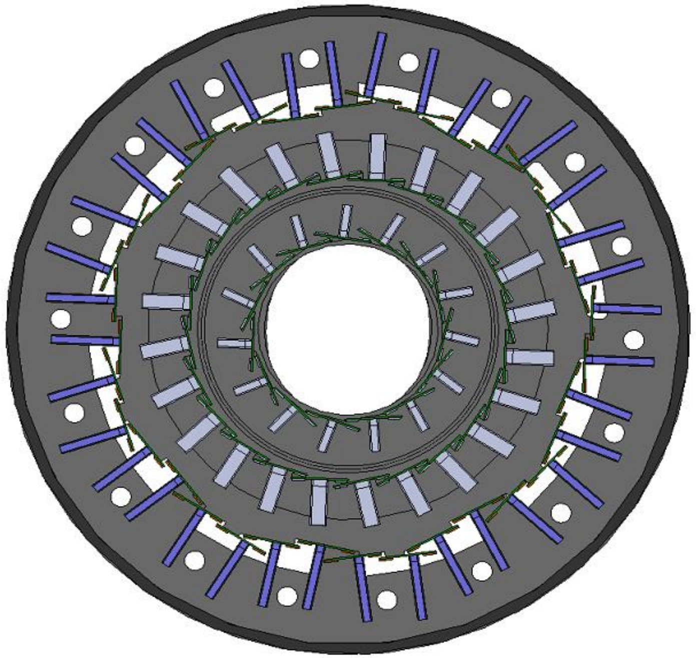
    - Sensor thickness: order of 50 μm → The smallest significant detail in the domain defines the minimum cell size for the mesh.

    - Staves size: order of 500 mm → Then, overall scale of the domain defines the maximum number of cells which are needed.

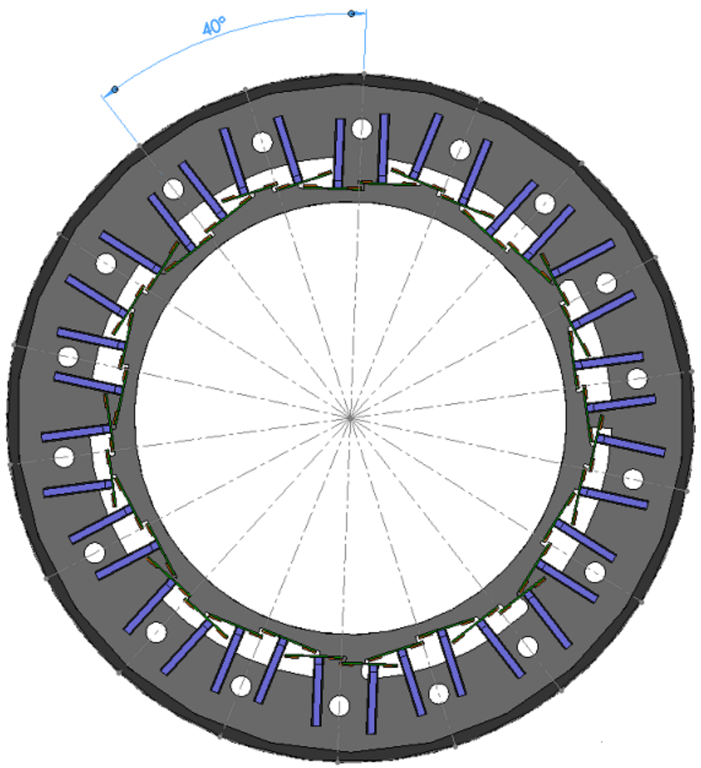
➢ **The number of cells needed to discretize the domain quickly becomes enormous !**

SIMPLIFICATIONS OF GEOMETRY

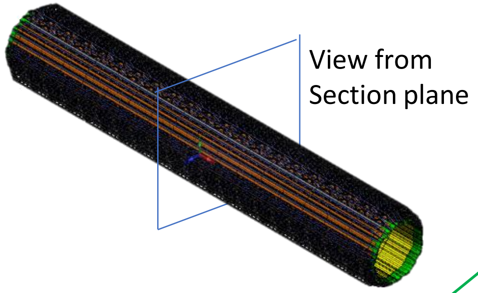
- Simplification of the geometry is needed. THE FIRST approach:

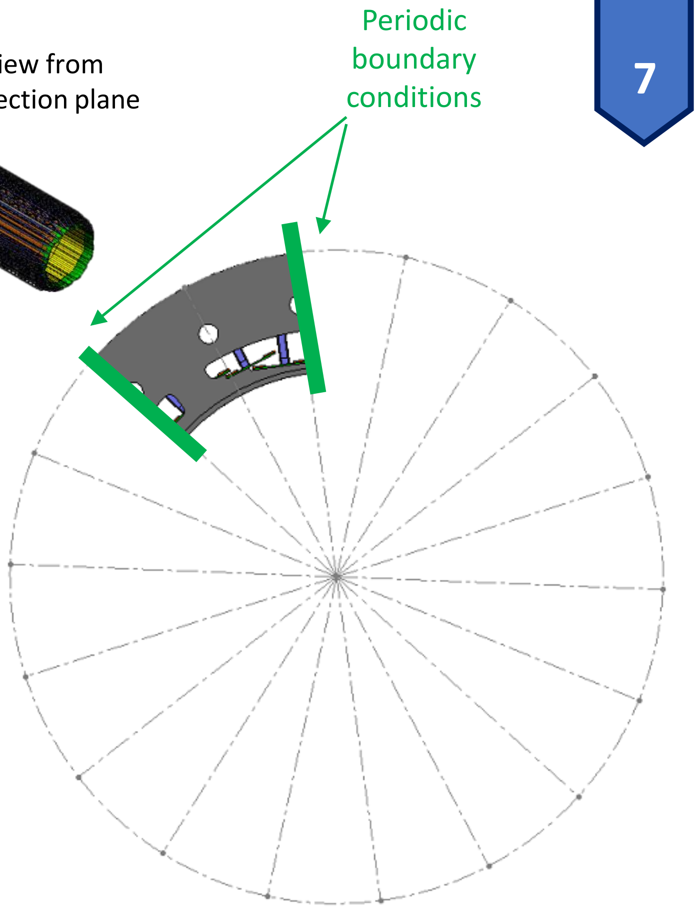


Starting from the full geometry



Removing layer 1 and 2



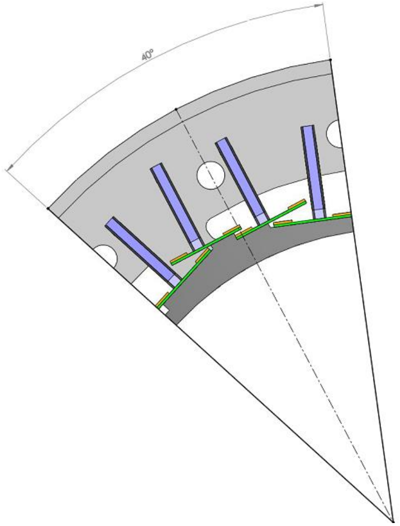


Focus on one sector of <u>Layer 3</u>
(40° circular symmetry)

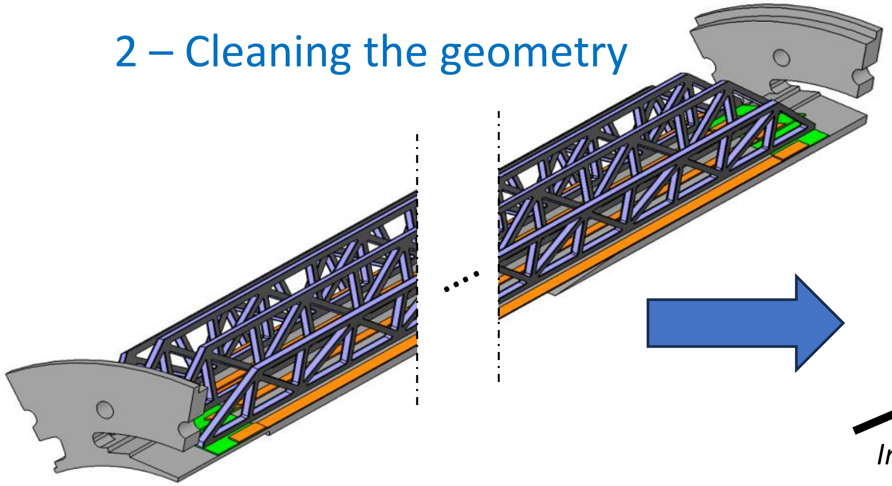
- A model for a 40° sector of layer 3 is managable with a medium-performance desktop workstation.
- Next target is to include also layer 2 and 1 (the layers are not airtight, air moves between layers)
- For simulating bigger domains, exploring higher performance machines or HPC capabilities.

STEPS FOR THERMAL SIMULATIONS

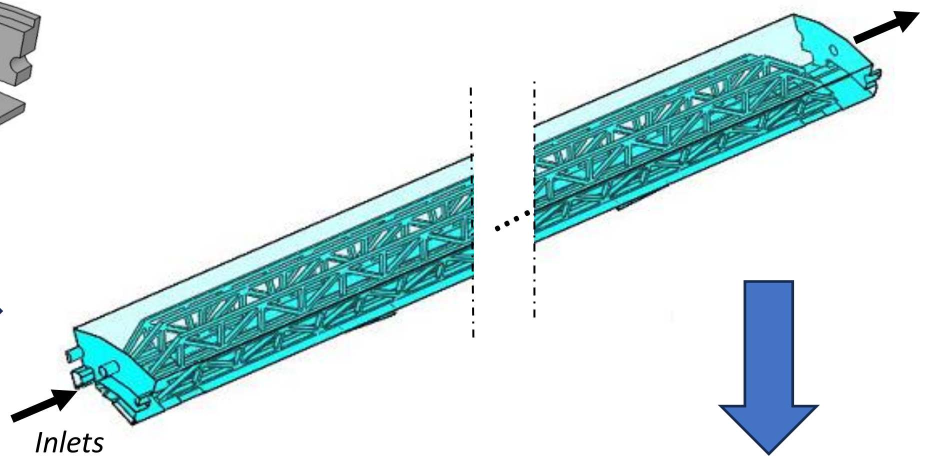
1 - Definition of the domain



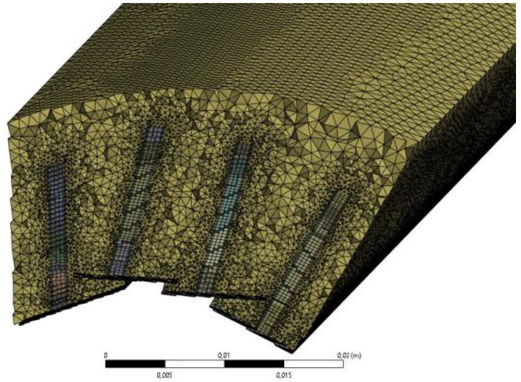
2 – Cleaning the geometry



3 - Adding the volume of fluid



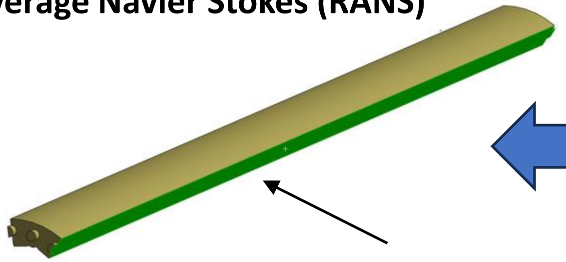
4 – Creating the grid



- Both the solid and fluid domains are simulated at the same time
  10’500’000 Elements
  17’000’000 Nodes

5 – Setting up the model

- ✓ Steady state conditions
- ✓ **Turbulent flow: Reynolds Average Navier Stokes (RANS)**
- ✓ Viscous model k-ω SST
- ✓ Pressure based solver
- ✓ No radiation heat exchange
- ✓ Volumetric constant power source on sensors volume

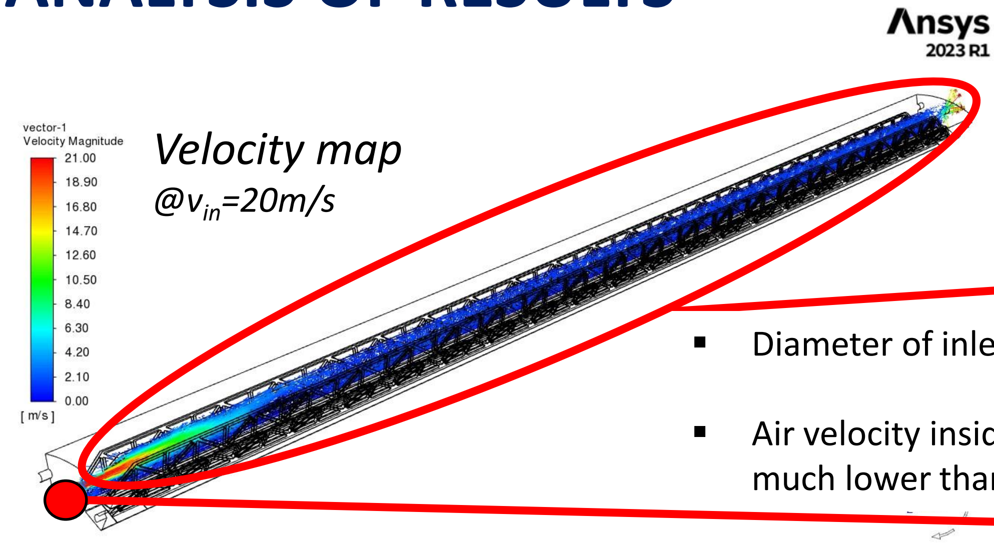


6 – Run calculations

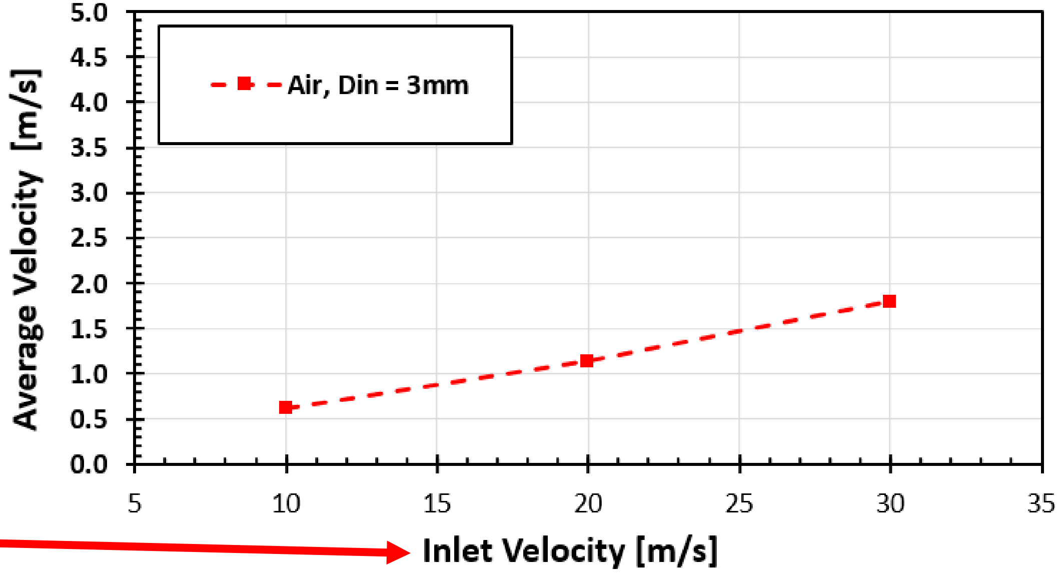
We used: i9-10900X CPU, 20 threads, RAM 128 GB, 2 TB SSD

ANALYSIS OF RESULTS

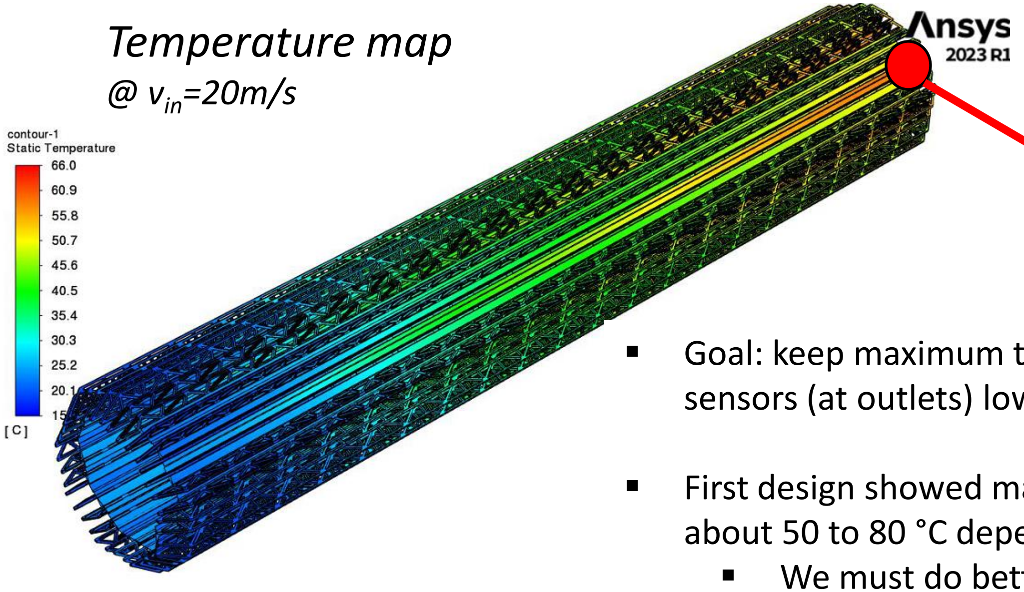
Velocity map
@v_{in}=20m/s



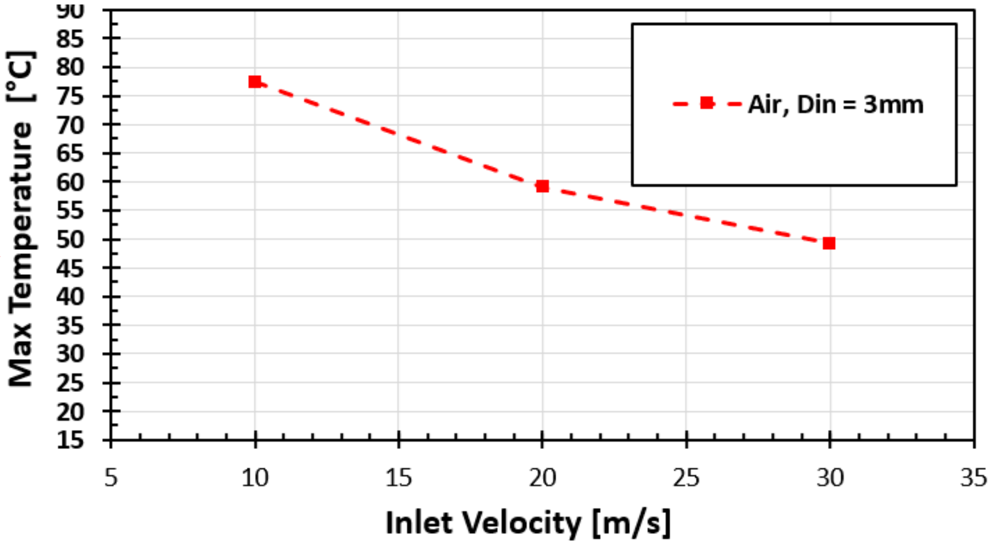
- Diameter of inlet holes D_{in} = 3mm
- Air velocity inside the cylinder much lower than inlets.



Temperature map
@ v_{in}=20m/s

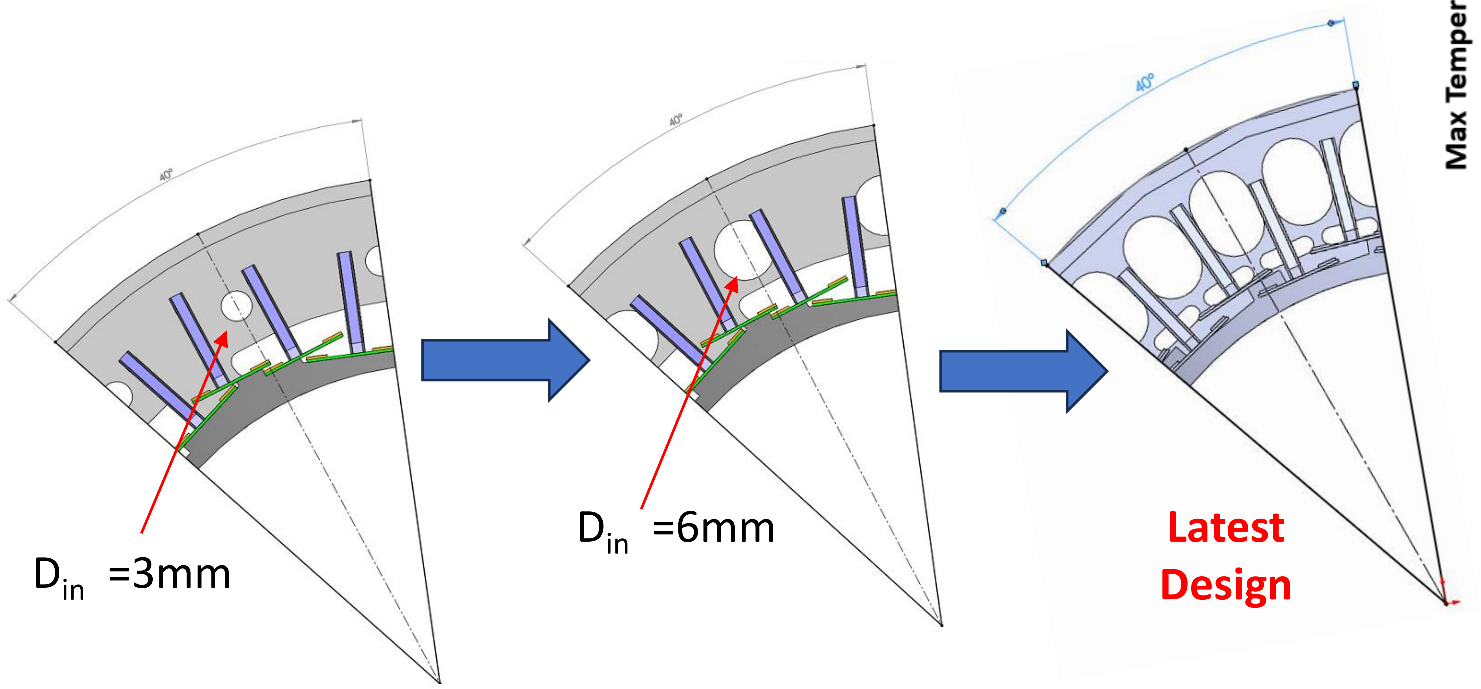


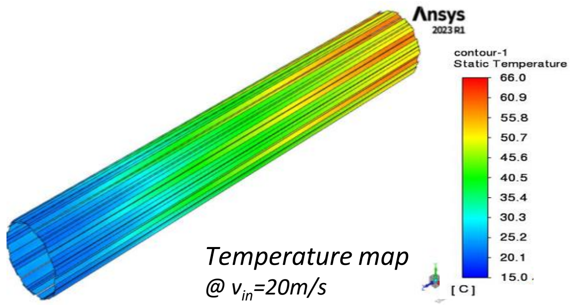
- Goal: keep maximum temperature of sensors (at outlets) low.
- First design showed max temperature about 50 to 80 °C depending on air flow.
  - We must do better!

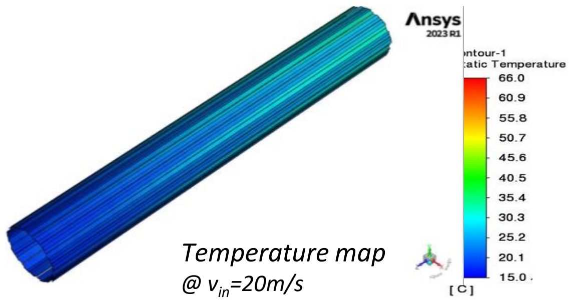


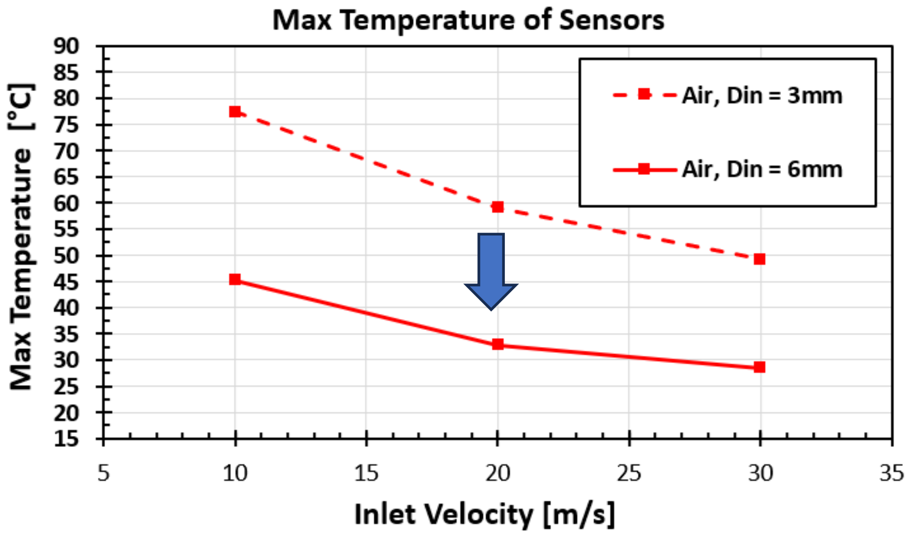
DESIGN OPTIMIZATION

- Big improvement just increasing the diameter for inlets holes.
  - Mass flow rate increase as well (for same v_{in}).







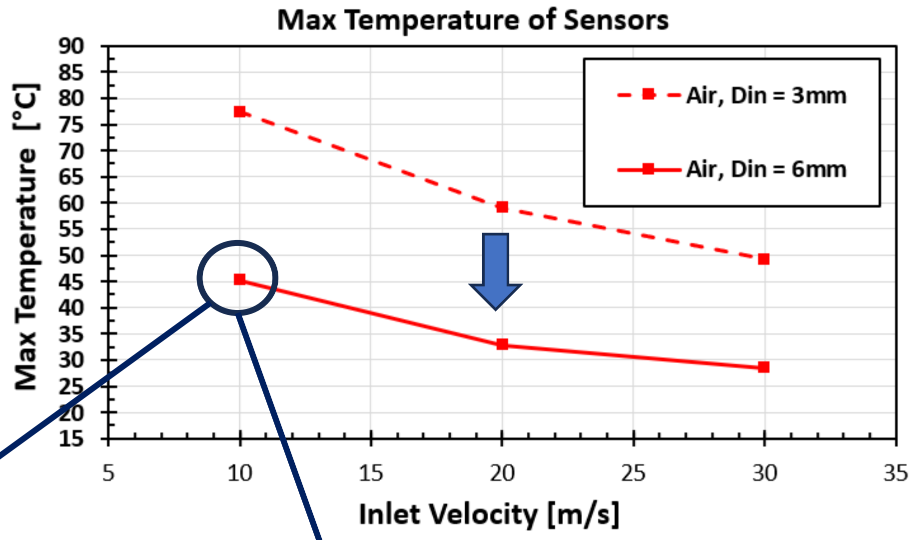


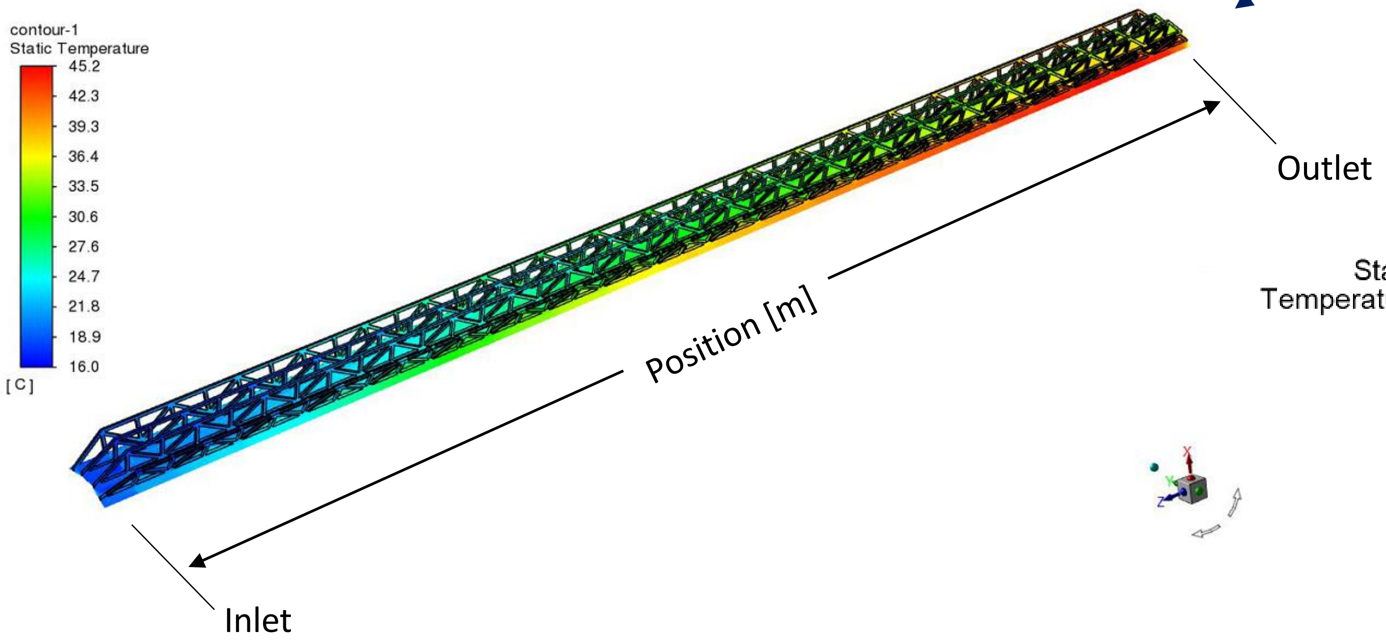
- Obtained max sensor ΔT wrt inlet air temp. about 15 to 30°C. (Function of air velocity)

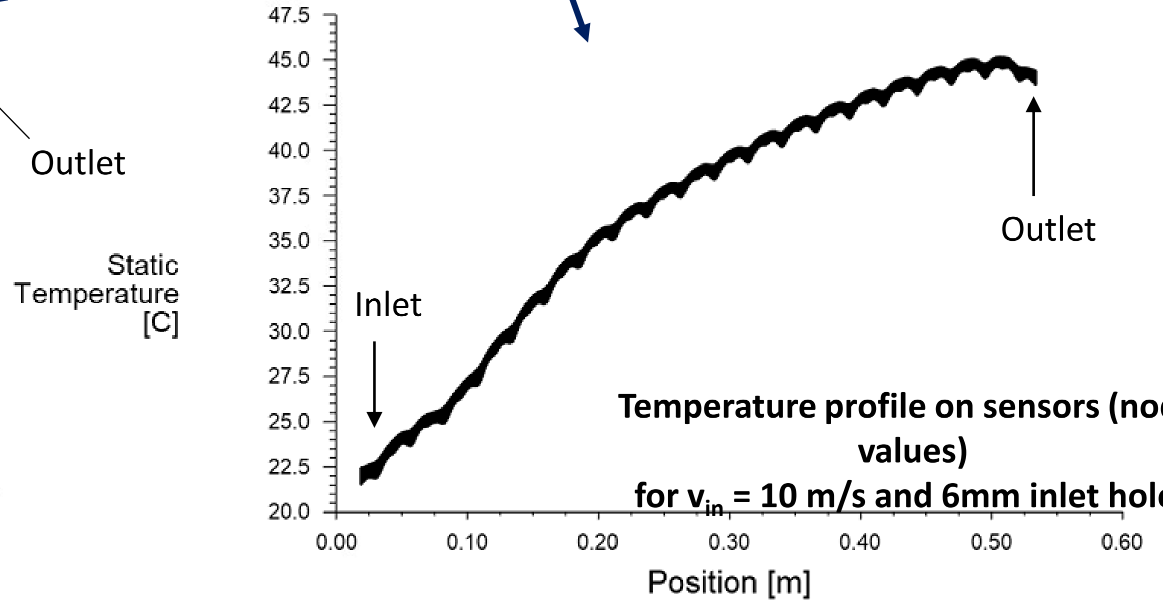
DESIGN OPTIMIZATION

- Big improvement just **increasing the diameter** for inlets holes.
  - Mass flow rate increase as well (for same v_{in}).

- Check also the max temperature gradient allowed by sensors along the stave:
  - Now worst case about **22°C** for V_{in}= 10m/s
  - Margin for improvements
  - Already better values for higher velocities





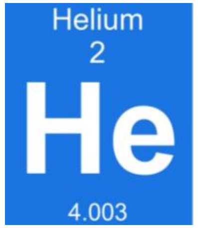


Temperature profile on sensors (nodal values) for v_{in} = 10 m/s and 6mm inlet holes

ANOTHER POSSIBILITY: CHANGING THE FLUID PROPERTIES

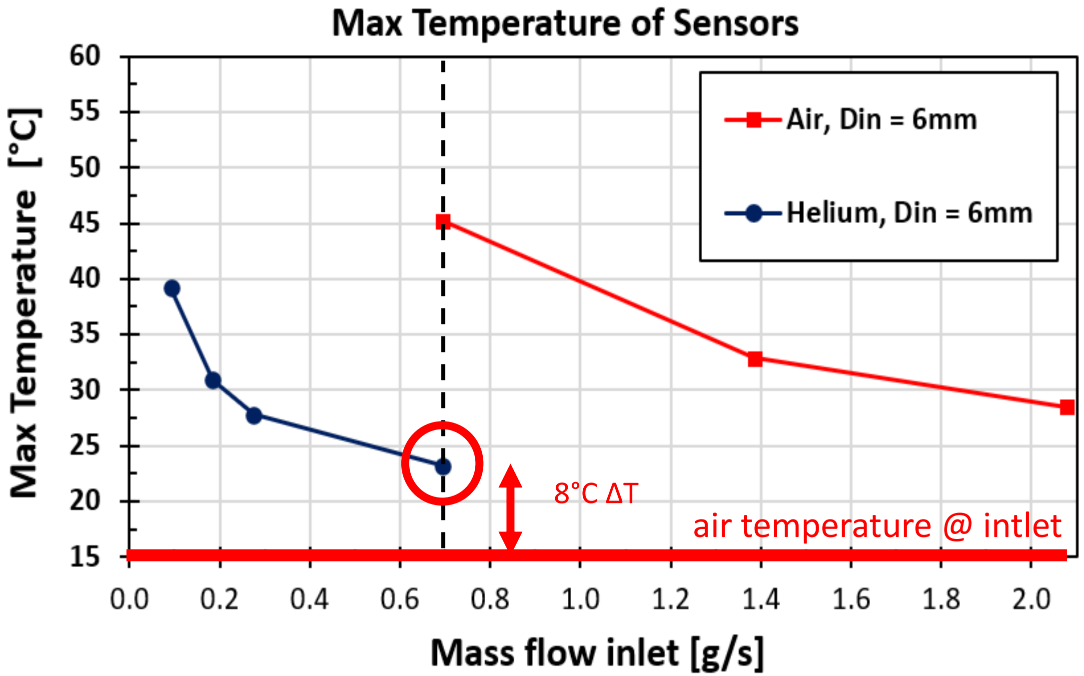
- The **use of helium** as a fluid instead of air was investigated.

- Properties of helium taken from Ansys libraries:
  - Lower density than air.
  - Higher thermal conductivity than air.



	Air	Helium
Density [kg/m³]	1.225	0.1625
Specific Heat [J/(kg K)]	1006	5193
Thermal Conductivity [W/(m K)]	0.024	0.152
Viscosity [kg/(m s)]	1.79e-05	1.99e-05

$$\dot{m} = \rho \cdot \boldsymbol{v} \cdot \boldsymbol{A}$$

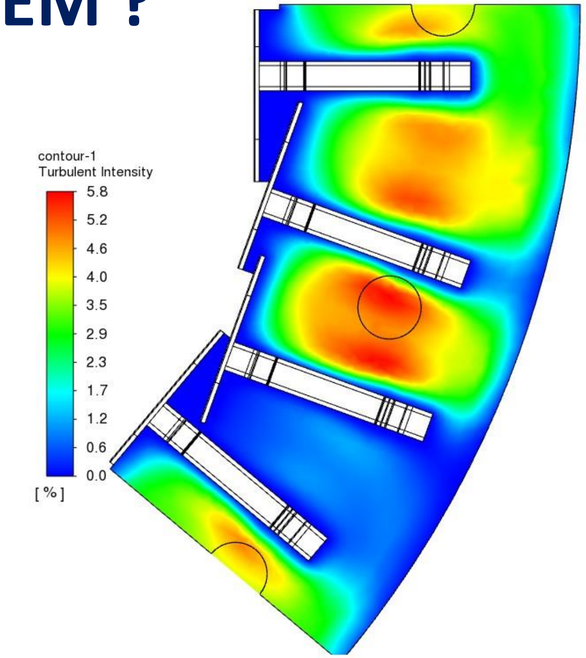


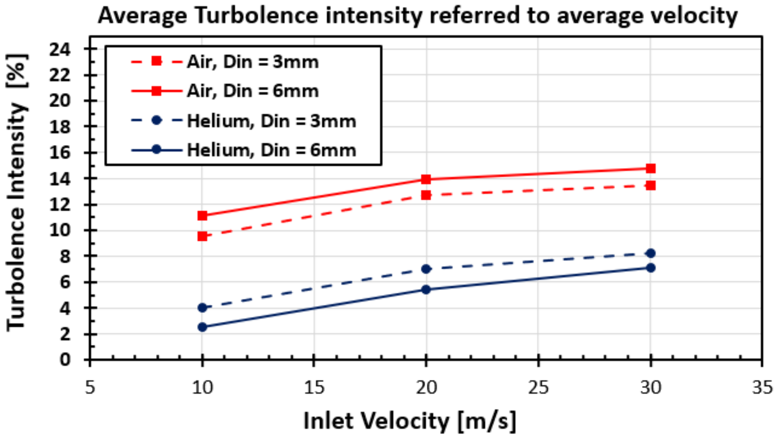
- Comparison is interestesting considering the same mass flow rate instead of same inlet velocity.

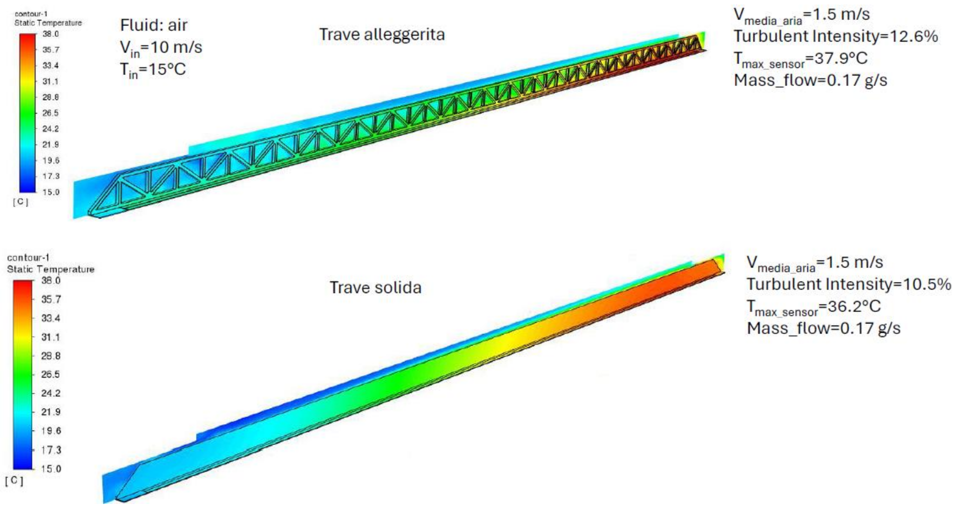
- Helium offers much better cooling than air for the same mass flow rate.

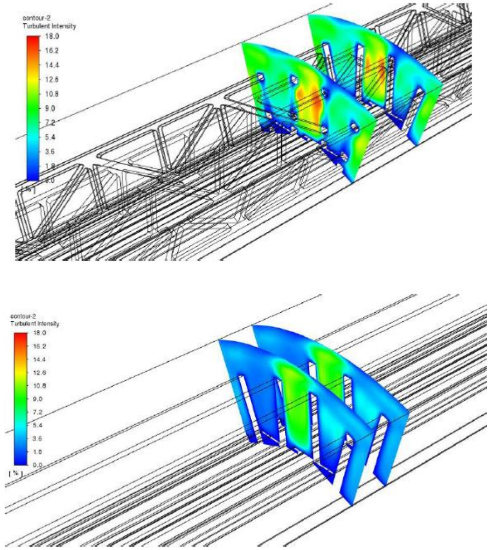
WHAT IS MAXIMUM AIRFLOW TOLERATED BY THE SYSTEM ?

- The **turbulence** generates a mixing of air which favors heat exchange.
- But also induces vibrations on the lightweight structure.
- Turbulence is generated by the air moving through the stave holes, but also depends on the air injection system (the compressor for example).
  - ➢ **Turbulence at the inlet is another input for the model**
    (Current Hypothesis 5 to 10%)
- Need to study the **fluid-induced-vibrations to the structure**



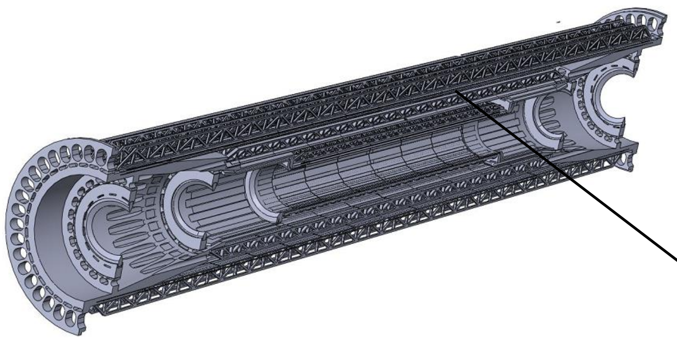






A MULTIPHYSICS MODEL FOR THE EVALUATION OF MECHANICAL STRESS

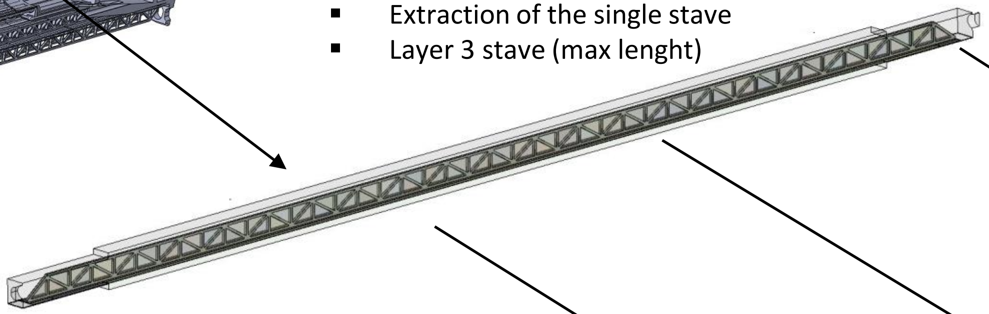
- Define a tool to evaluate whether the air flow necessary to remove the heat generates excessive vibrations on the stave.

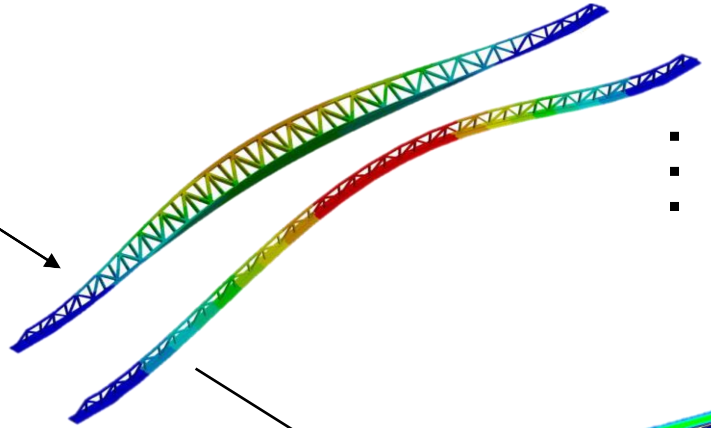


CAD Model

Geometry simplification

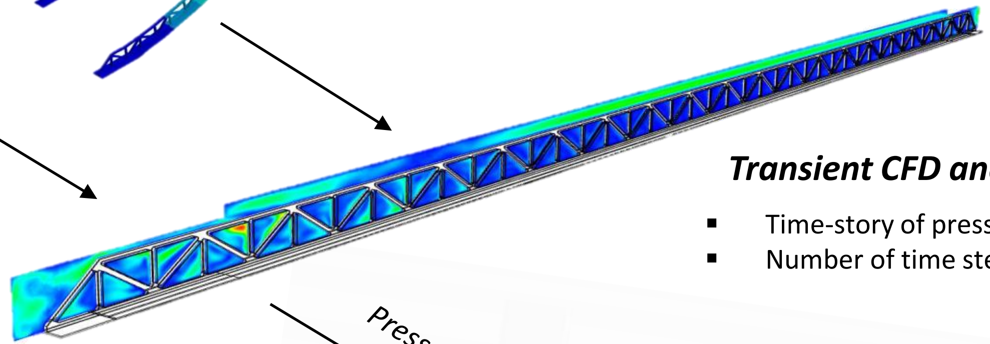
- Extraction of the single stave
- Layer 3 stave (max lenght)





Modal analysis

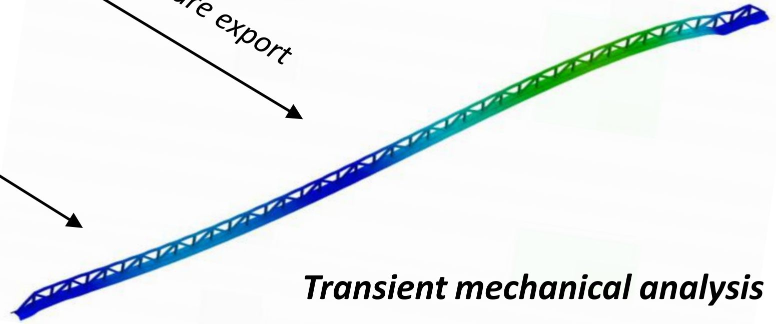
- Natural frequencies.
- Participation factors.
- Define time step for transient analisys.



Transient CFD analysis

- Time-story of pressure load.
- Number of time steps.

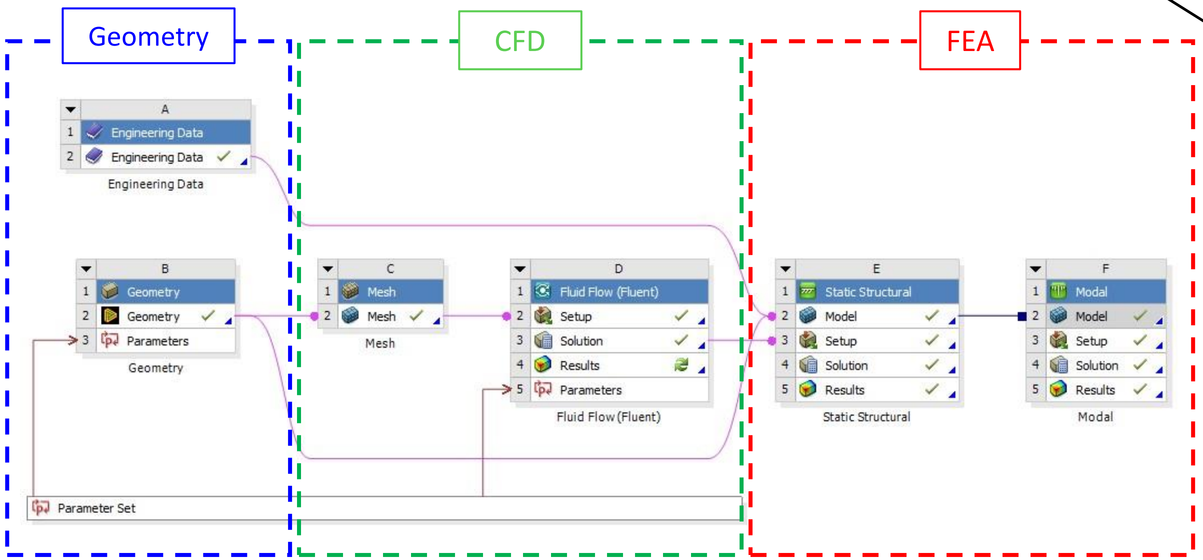
Pressure export



Transient mechanical analysis

- Dynamic displacement

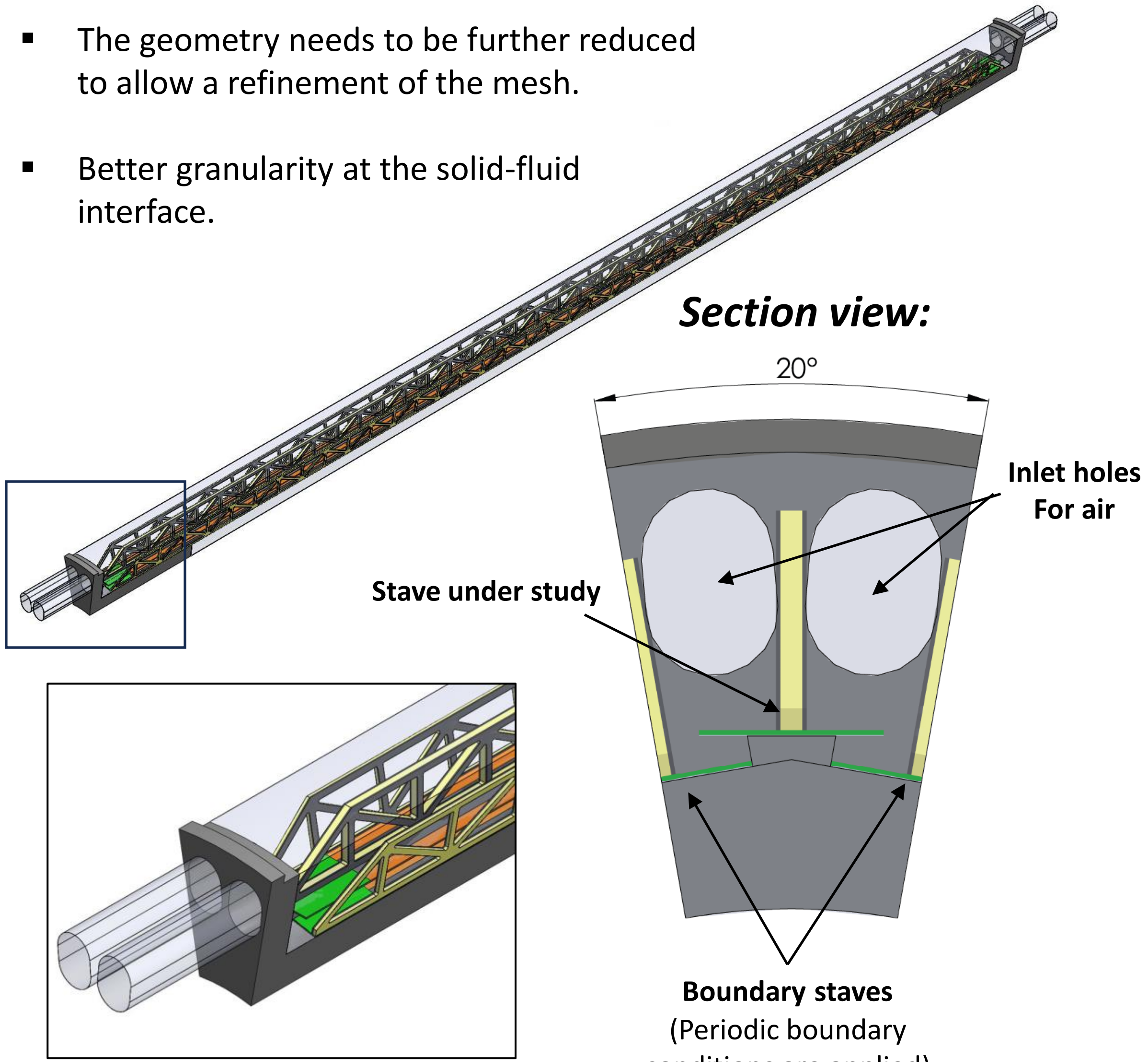
- Flow of data managed by Ansys Workbech



GEOMETRY

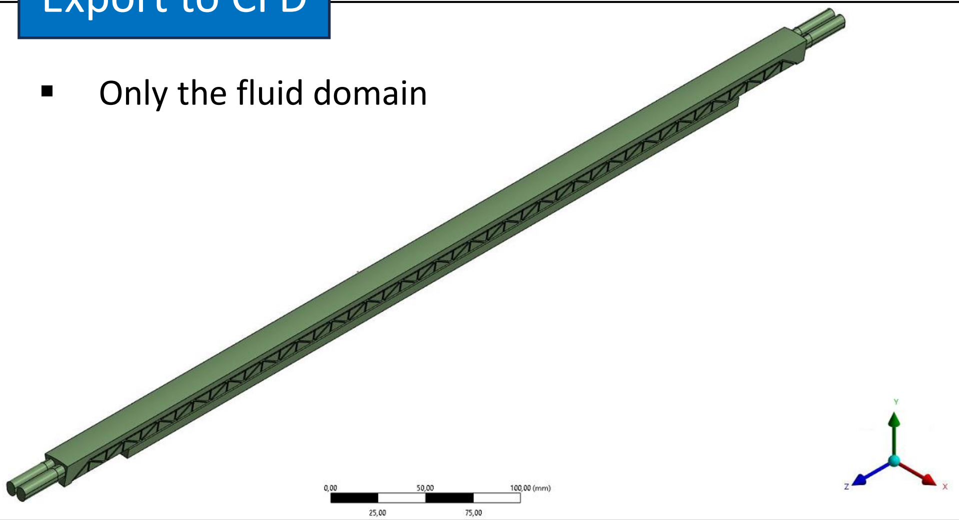
- The geometry needs to be further reduced to allow a refinement of the mesh.
- Better granularity at the solid-fluid interface.

Section view:



Export to CFD

- Only the fluid domain



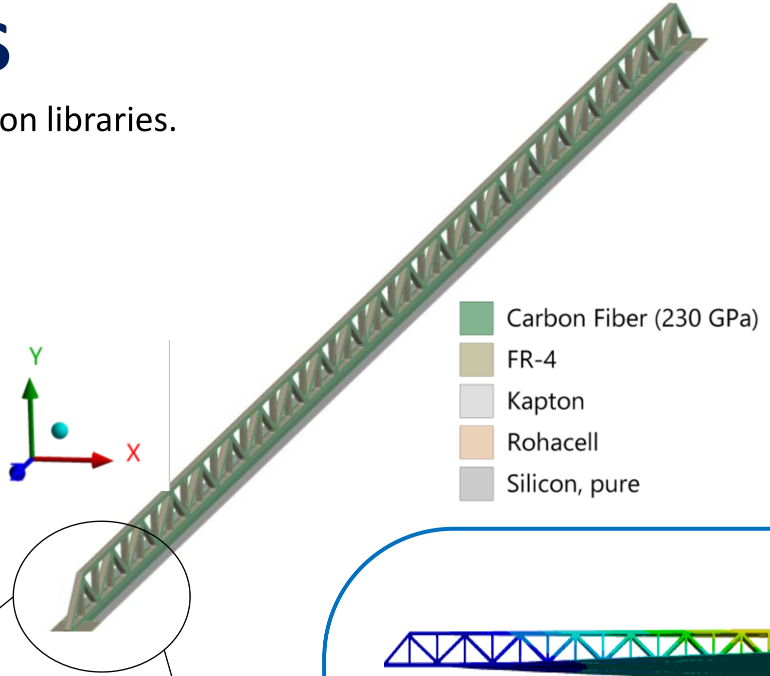
Export to FEA

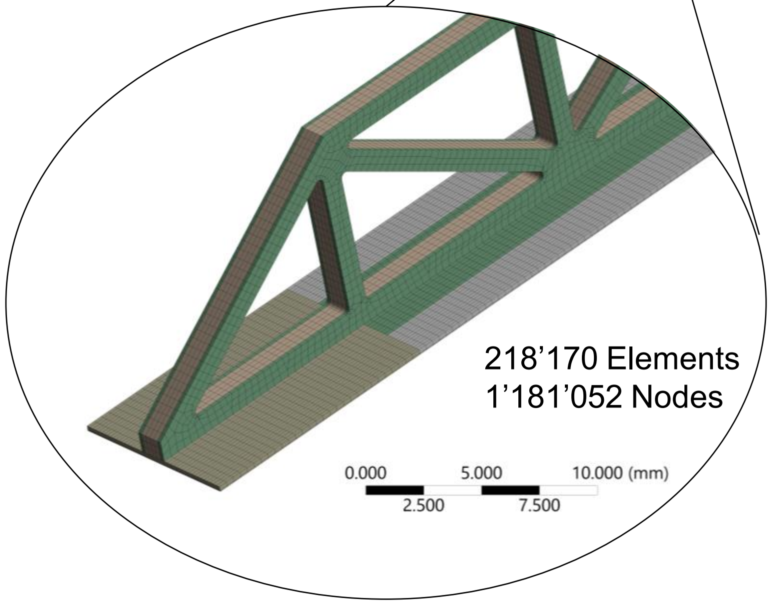
- Only the central stave

MODAL ANALYSIS

- Material data taken from common libraries.
- Frequencies > 100 Hz
- 1° Mode torsional
- Natural frequencies are used to define the time step for next transient analysis:

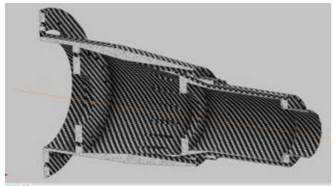
$$\Delta t < \frac{1}{f_{max} \cdot 10}$$

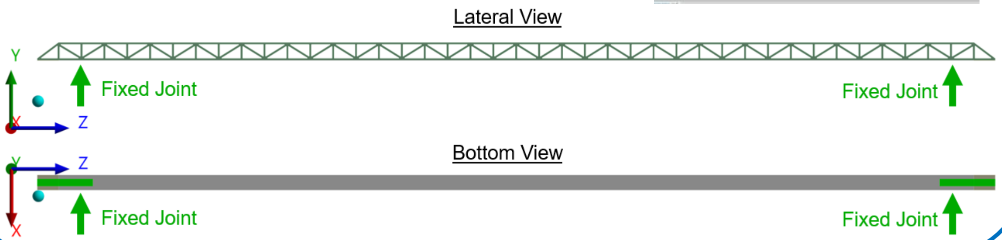




Constraints

- Hypothesis: support cone (anchoring of each stave) not deforming with time.

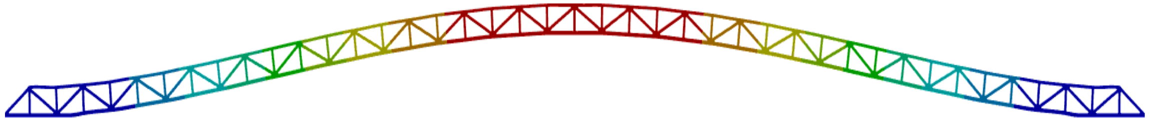




Results

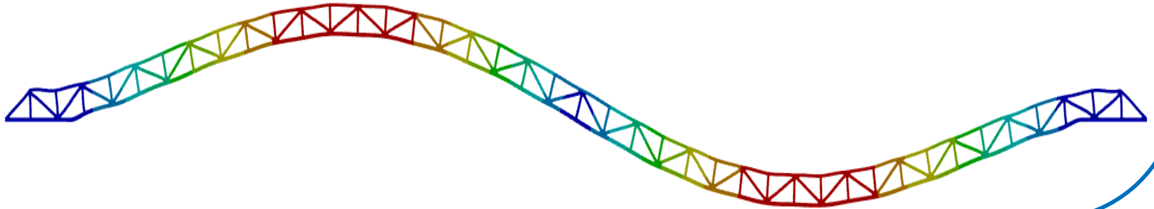
Mode	Frequency [Hz]
1	118.8

Mode	Frequency [Hz]
2	147.1



Mode	Frequency [Hz]
4	292.1

Mode	Frequency [Hz]
5	394.4

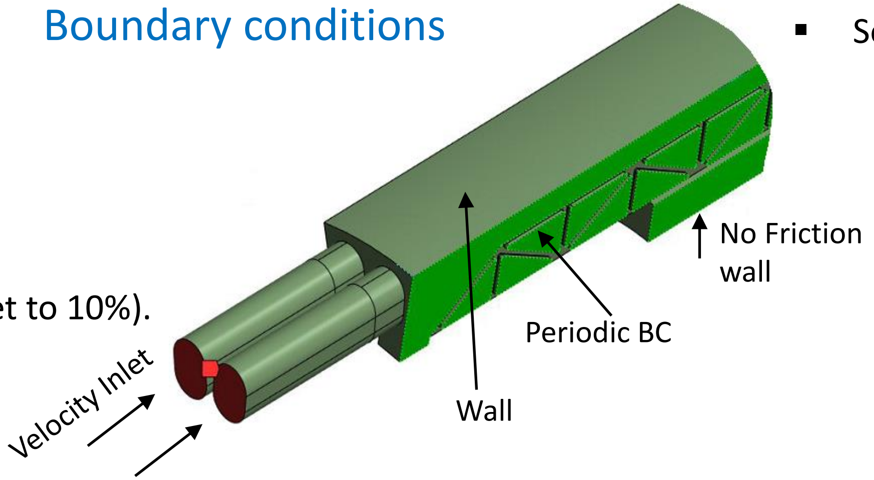


CFD TRANSIENT ANALYSIS

- Re > 4'000 – Turbulent flow $Re = \frac{F_{inertia}}{F_{viscous}} = \frac{\rho V D_H}{\mu}$
- Viscous model: Large Eddy Simulation (LES) Model
- Average velocity at inlet: V_{in_avg} = 10 m/s
- Turbulence generator at inlet (turbulence intensity set to 10%).
  - Spectral Synthesizer Algorithm [

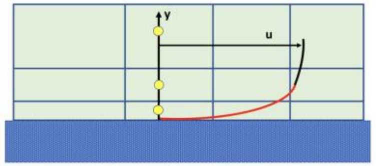
$$u'(\mathbf{r},t)=2K_t\sum_{n=1}^{N}\sqrt{q^n}\left[\boldsymbol{\sigma}^n\cos\left(k^n\mathbf{d}^n\cdot\mathbf{r}+\phi^n+s^n\frac{t}{\tau}\right)\right]$$

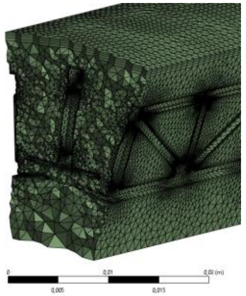
Boundary conditions



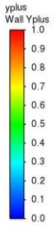
- Solver parameters:
  - Time step = 6.8e-4 s
  - N° of time steps (Total simulated time) defined by the time a flow particle takes to cross the entire beam, given the RMS velocity of the flow along the axis.
  - NOTE: this involve long calculation time !

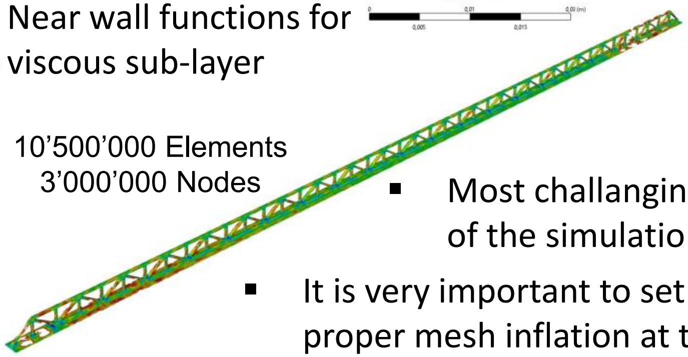
Mesh





- Near wall functions for viscous sub-layer

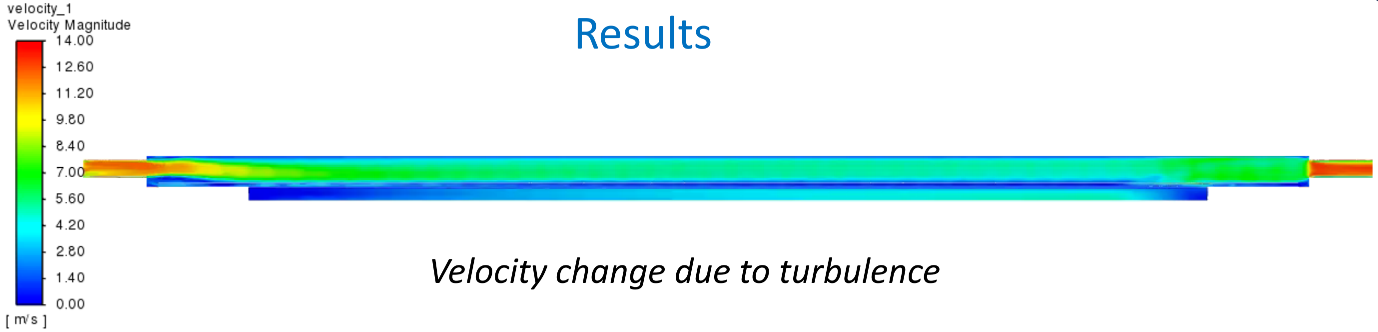




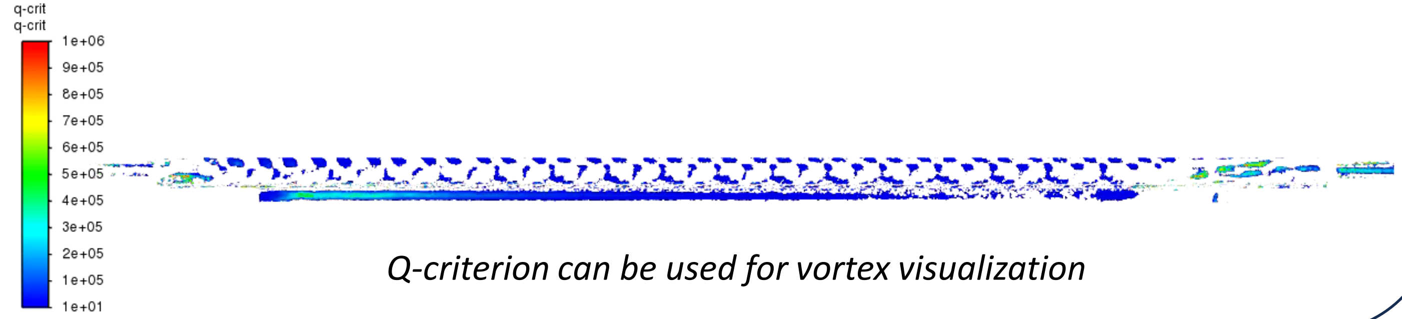
10'500'000 Elements
3'000'000 Nodes

- Most challanging part of the simulation.
- It is very important to set a proper mesh inflation at the boundary layers.

Results



Velocity change due to turbulence

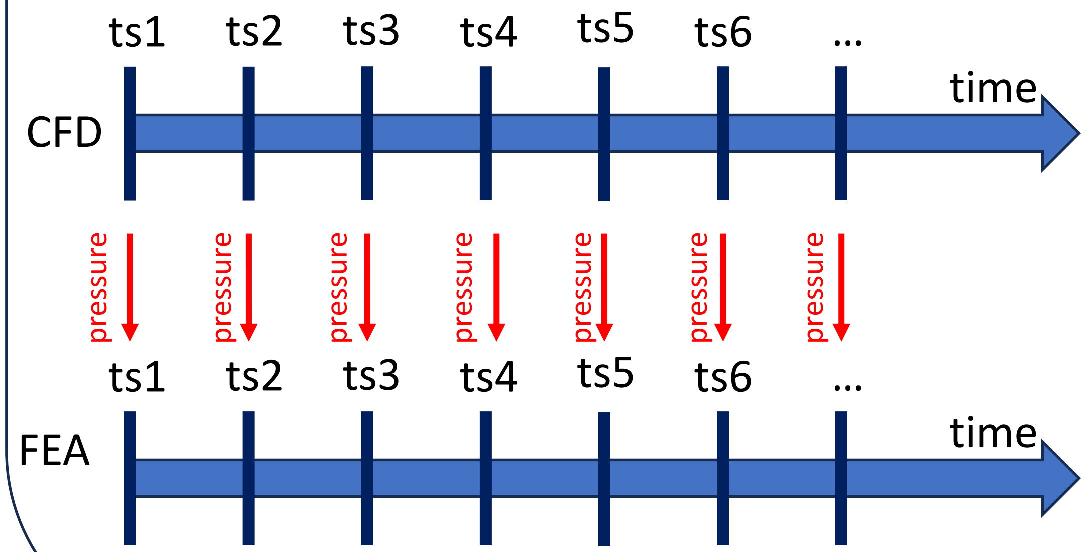


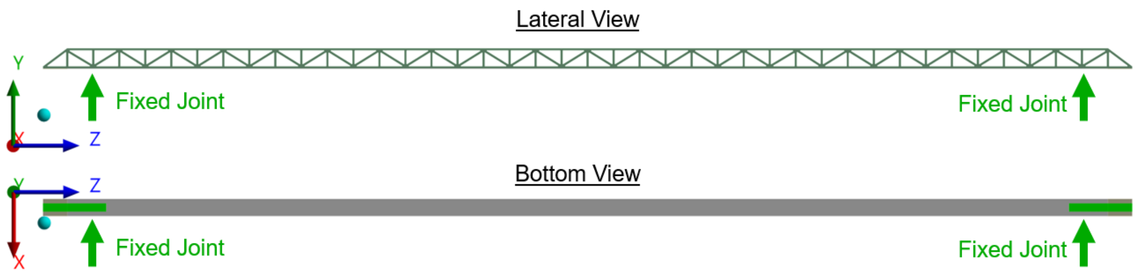
Q-criterion can be used for vortex visualization

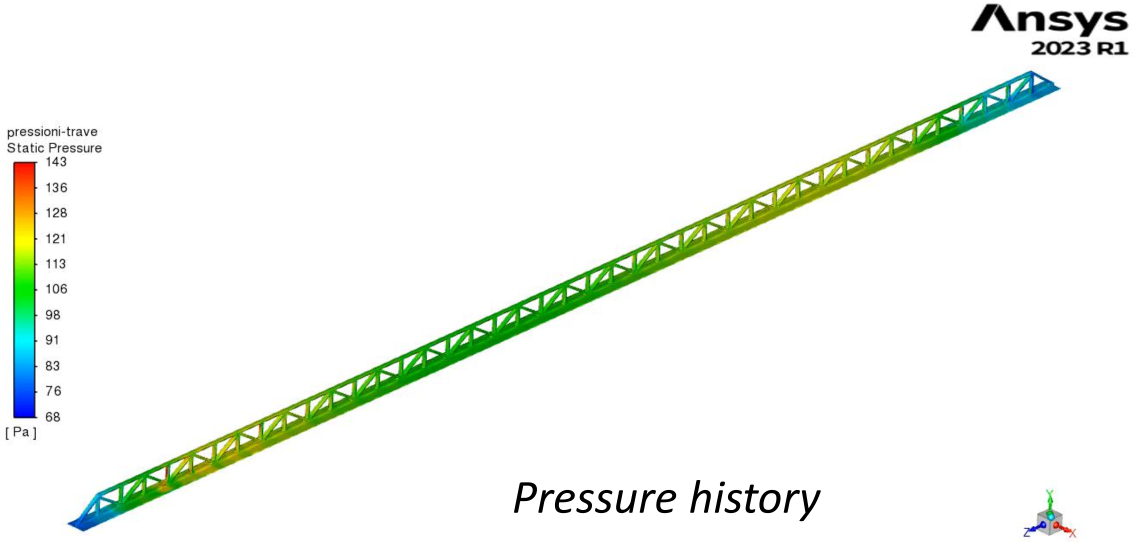
SETTING THE MECHANICAL TRANSIENT ANALYSIS

Boundary conditions

- Same constraints and materials used for Modal analysis.
- Pressure history is imported from CFD as external load on all the boundary surfaces of the beam.
- Set time step (ts) for FEA simulations the same as for CFD (pressure load is updated each time step).

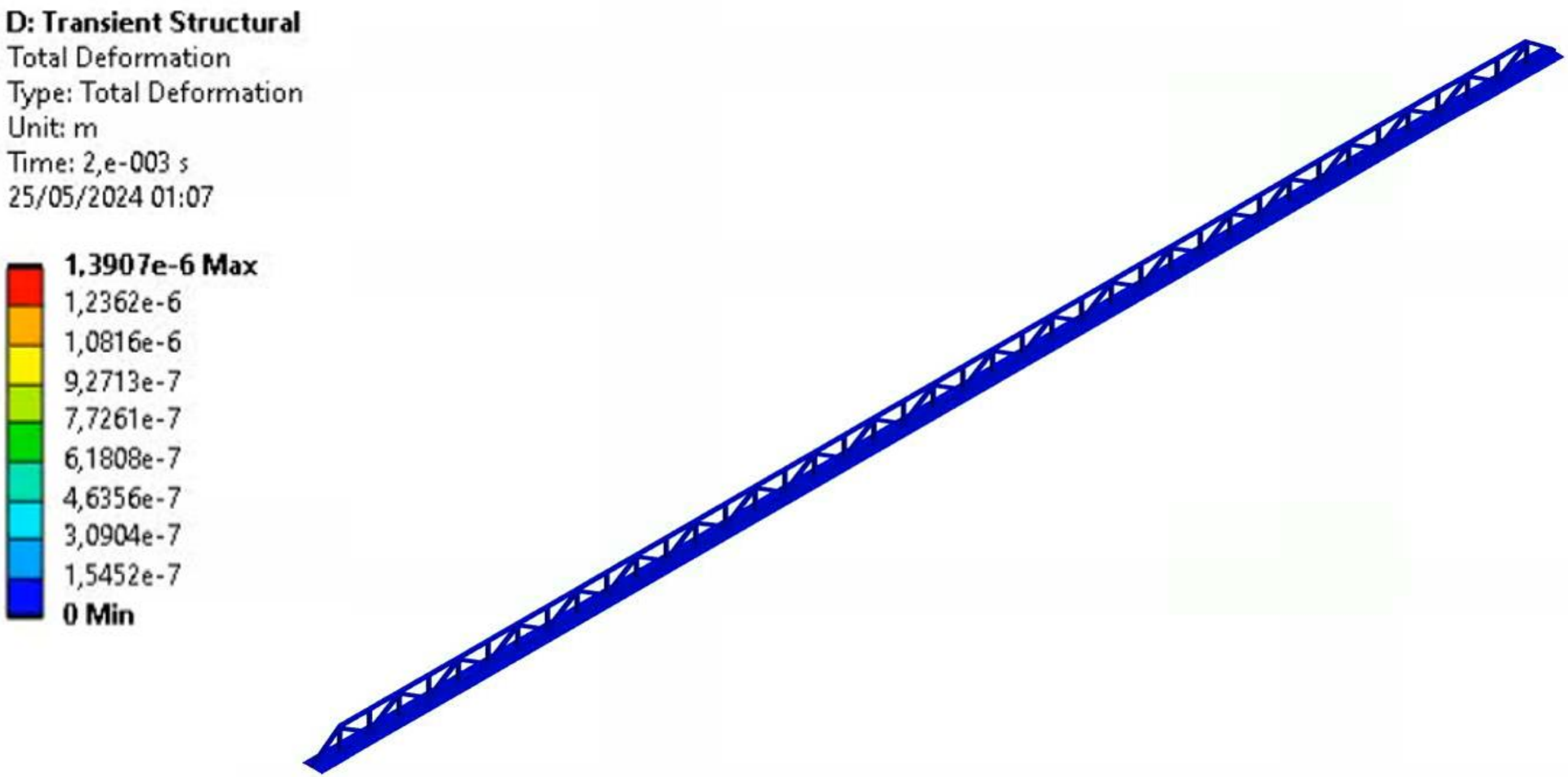






Pressure history

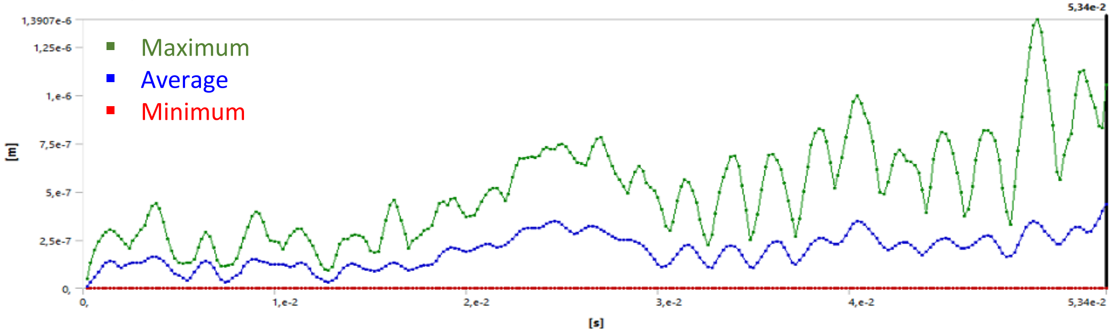
RESULTS


D: Transient Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 2,e-003 s
25/05/2024 01:07
1,3907e-6 Max
1,2362e-6
1,0816e-6
9,2713e-7
7,7261e-7
6,1808e-7
4,6356e-7
3,0904e-7
1,5452e-7
0 Min

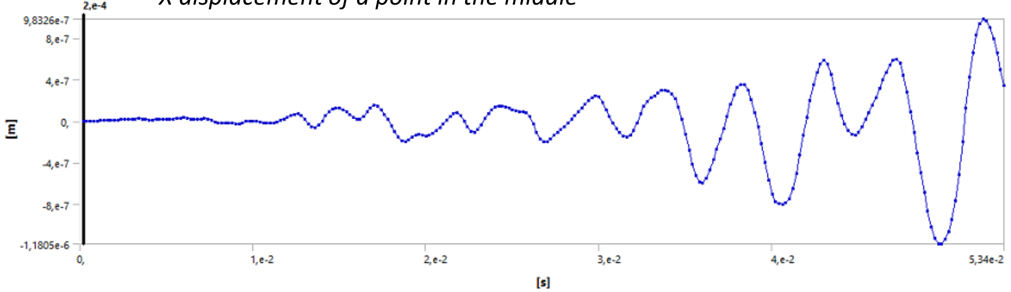
RESULTS

- Simulation takes some time steps before first modes are fully developed.
- First two natural frequencies are clearly triggered.

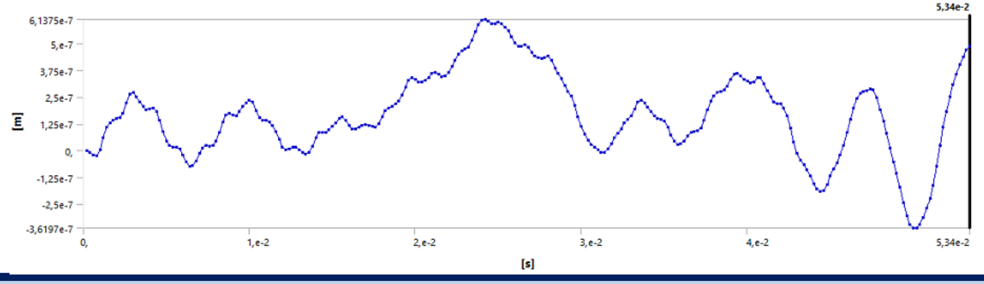
Global deformation:

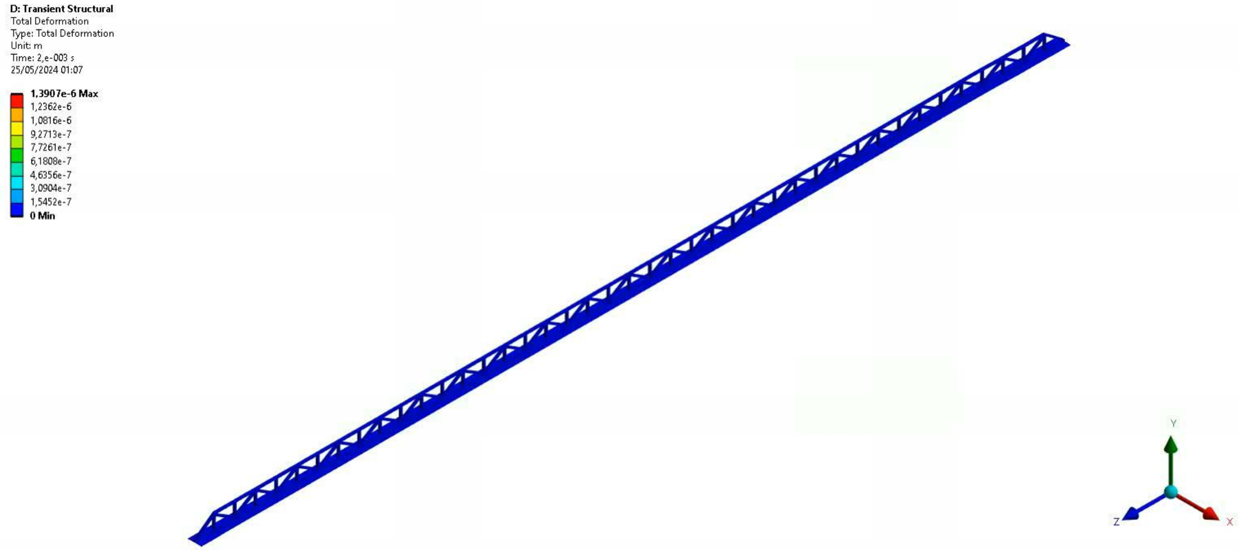


X displacement of a point in the middle

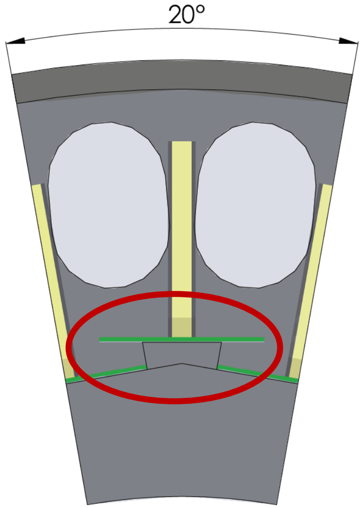


Y displacement of a point in the middle

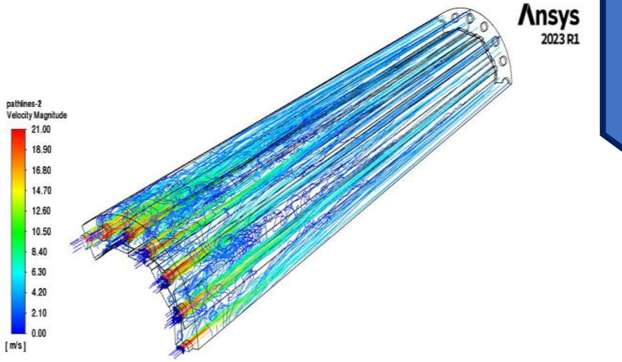




- Maximum displacement magnitude about 1.5 µm @ v_{in} = 10m/s
  - Mainly due to first-mode
  - This give hint to improve stiffness in the fixation region to the support cone.
- Model must be tuned with experimental data.
  - Look for suitable displacement sensors for the micron scale!



SUMMARY & FINAL REMARKS



- Numerical simulations are a valid tool for guiding the design of the vertex detector.
- A FVM model is useful for predicting the maximum temperature of the sensors.
  - Promising results on the simulated geometry with staves, still margin for improvements.
- A Multiphysics model joining CFD and FEA can be used to analyze the fluid-induced-vibrations on the structure.
  - For an air velocity of 10 m/s, simulated oscillations came in the micron scale in the simulated geometry.
- The proposed modeling workflow can be applied to any geometry of the vertex.
- **An experimental validation is crucial to tune the model ! Simulation work needs to go hand by hand with tests.**

NUMERICAL SIMULATIONS ⟷ **EXPERIMENTAL TEST**

- This is particularly true for the study of the dynamic response, since the turbulence model to be used really depends on the application.

Thanks for your attention

Contacts:

- fabrizio.palla@cern.ch
- giorgio.baldinelli@unipg.it
- cristiano.turrioni@pg.infn.it
- filippo.bosi@pi.infn.it
- giulia.pascoletti@unipg.it