ABY NARAYAMPARAMBIL – KTH ROYAL INSTITUTE OF TECHNOLOGY

THESIS WORK PRESENTATION

DESIGN OF FLOW GUIDING PARTS THROUGH CFD TOPOLOGY OPTIMISATION

SCANIA

Design flow guiding parts through CFD topology optimisation

Need:

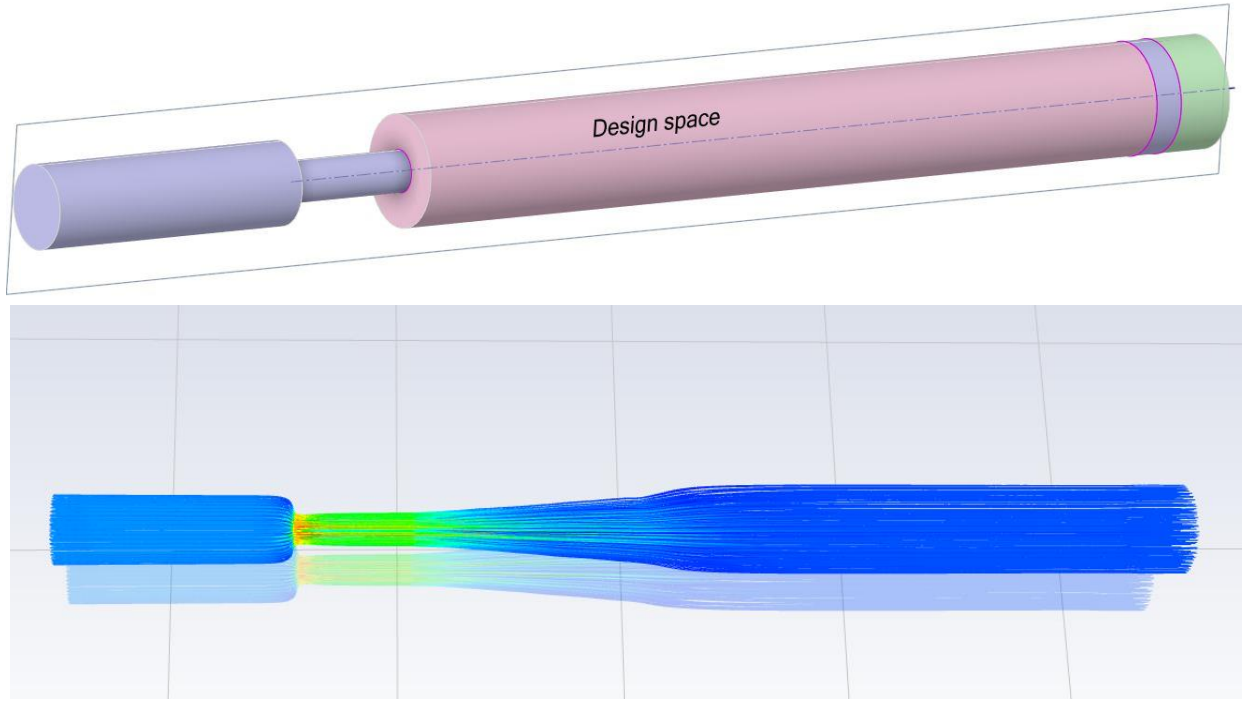
- Obtain a baseline design of flow guiding parts, from a pre-defined design space – helps designers to reduce time spent on iterative design loops
- Reduce backflow in flow sections – reduces pressure drop, consequently improves 'ui' in outlet sections

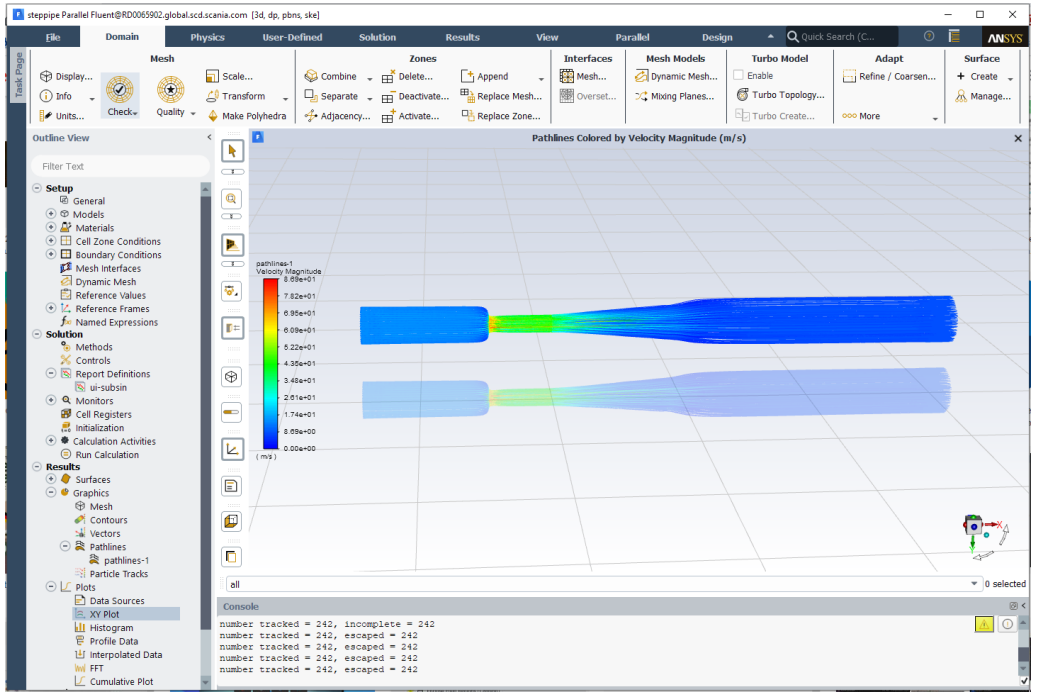
Goal:

- Evaluate flow topology optimisation software TOSCA Fluid in a case with catalytic substrate
- Evaluation based on: MTX model endplate

Working with TOSCA FLUID

Case: Stepped pipe with catalytic substrate





- Use tosca fluid gui to setup parameter file for fluid optimisation
- Submit job to the cluster
- Verify the results

Additional tasks done:

- Add a ui monitor and pressure drop monitor in fluent. These are not possible in tosca fluid 2019, not sure about later versions

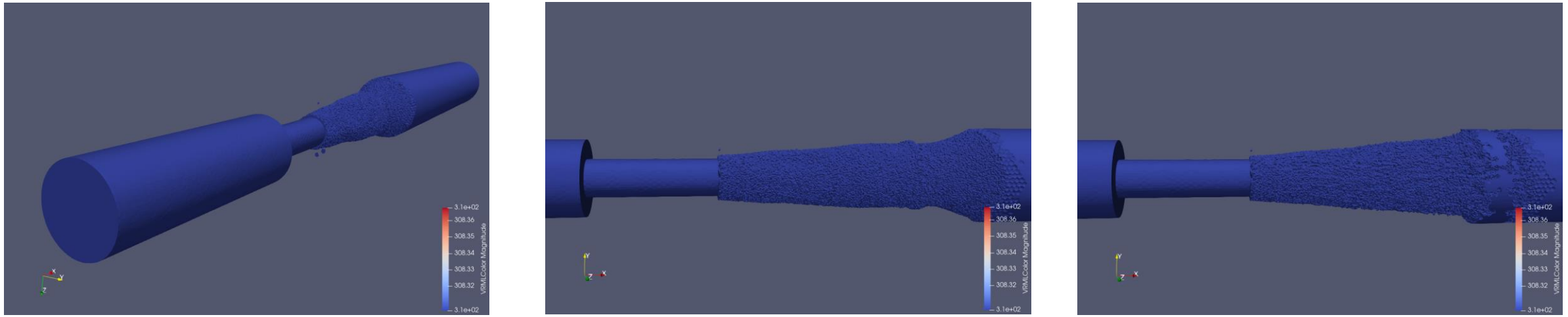
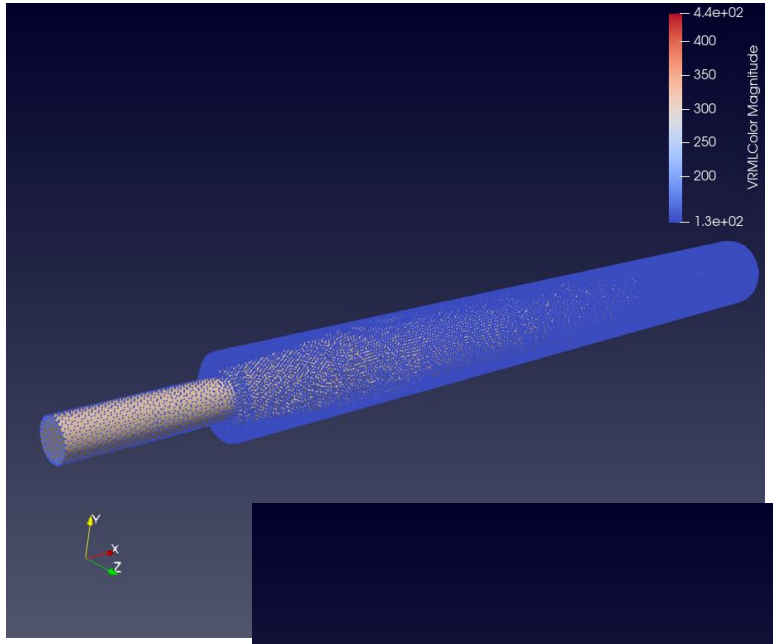
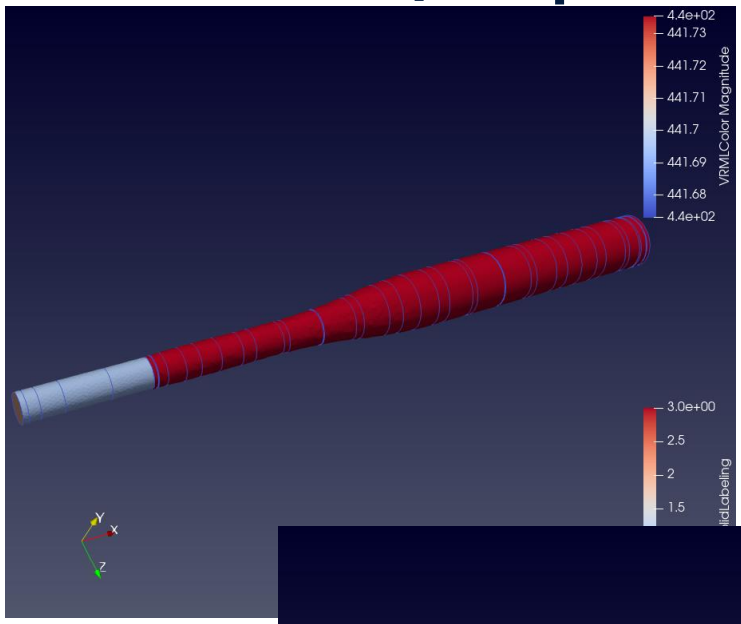
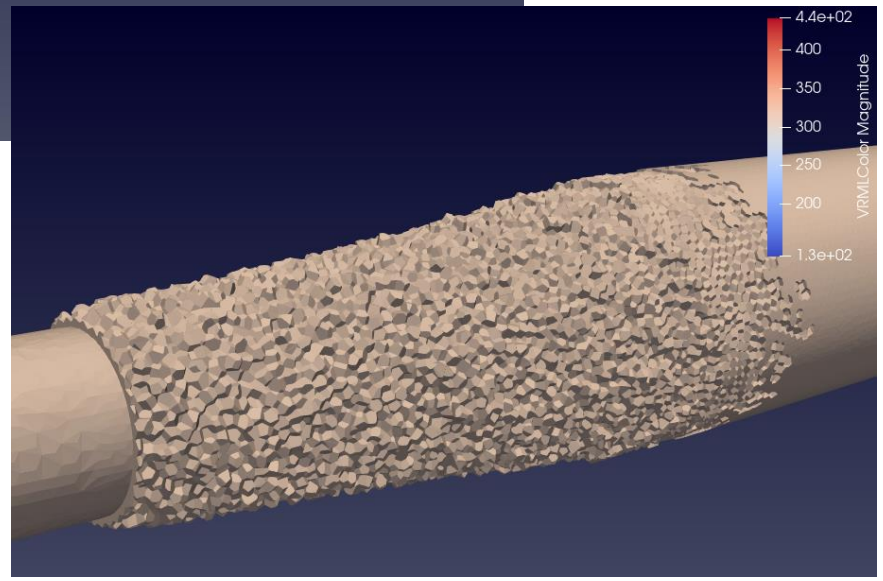


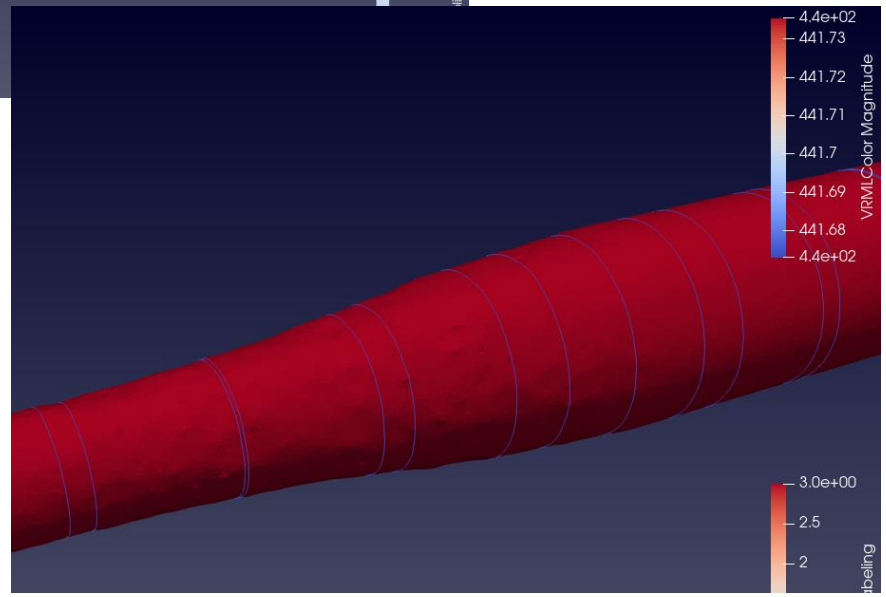
Figure: vrml file of optimised design after running tosca fluid (viewed on ParaView)

Case: Stepped pipe without catalytic substrate (Simpler case)









Assumptions when starting cases on tosca fluid

1. Optimised design would be similar for similar design spaces
2. Division of inlet and outlet interfaces of design space gives a better spread of particle track, between all the divided interfaces
3. Uniform flow over the design space’s outlet interface

Case: End plate

(90 degree elbow bent with different cross-sectional area at inlet and outlet, and a catalytic substrate in the flow section)

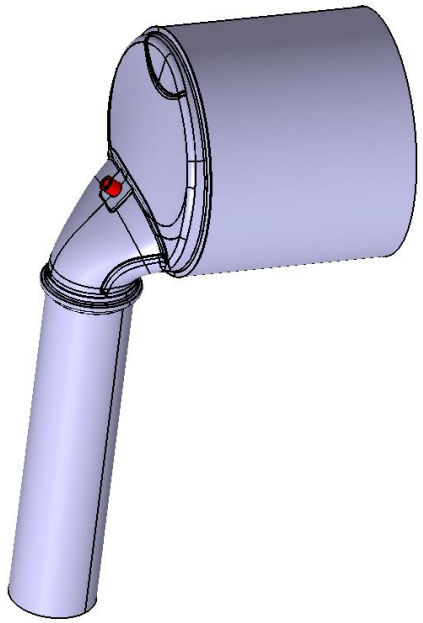
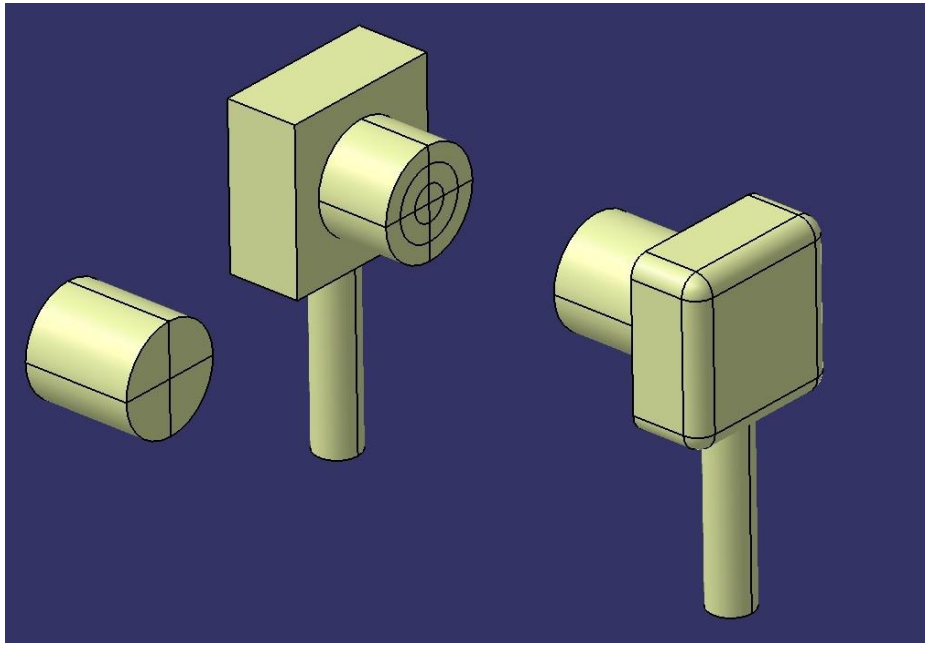
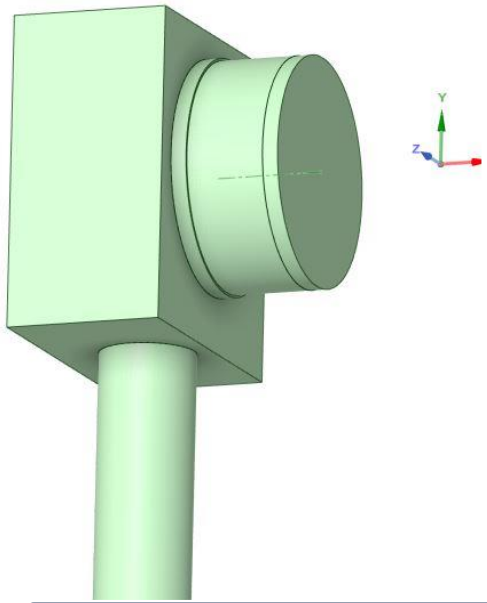


Figure: MTX endplate used for reference to model the design space and non-design space

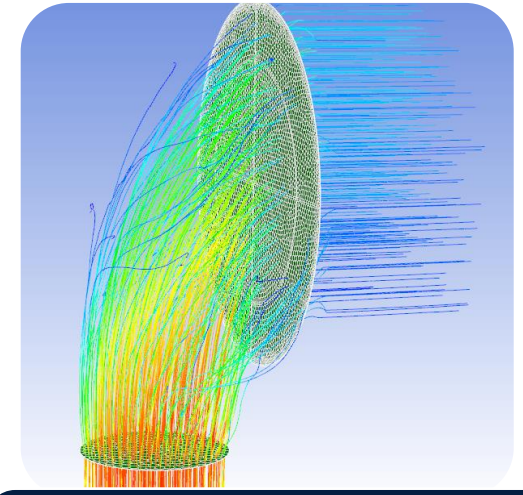
- Done on CATIA and saved as CATPart file.
- This is imported to spaceclaim for modifications, sharing the topology/surfaces, fixing issues in the model and refining it for Ansys fluent mesher.



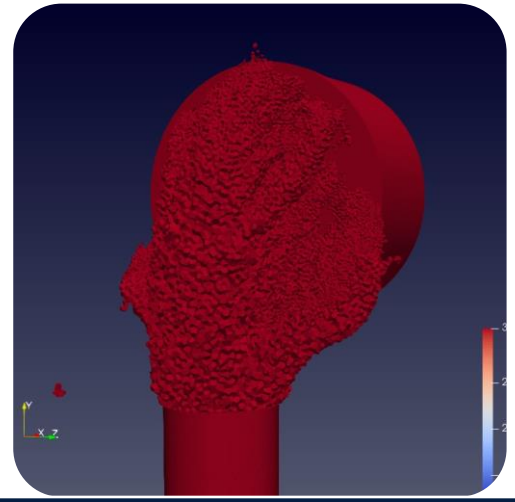
Workflow overview



Modelling fluid-flow region

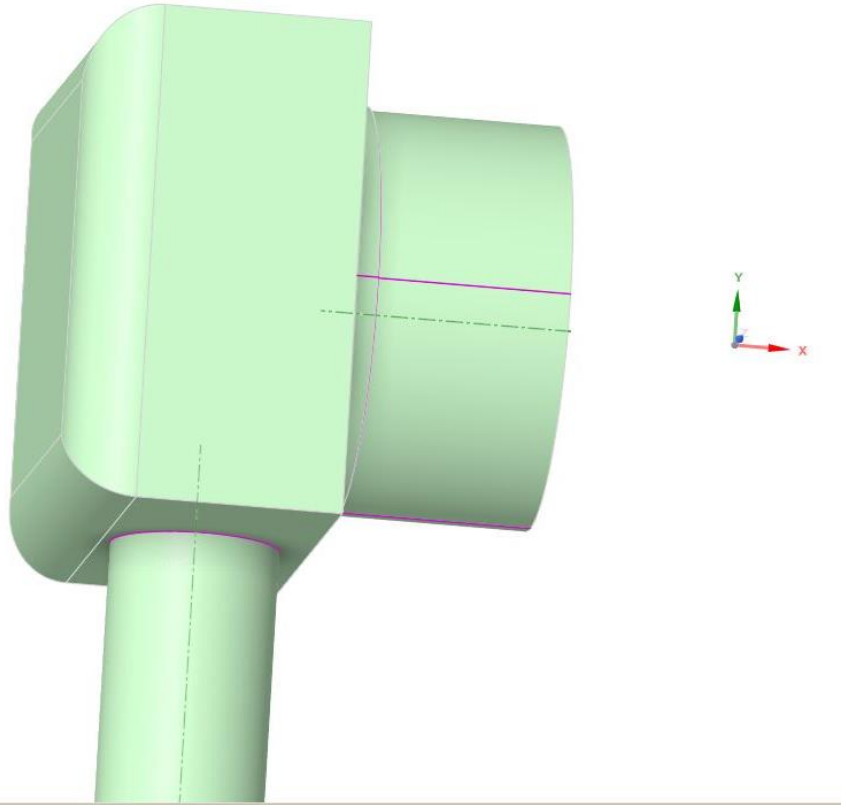


Ansys fluent meshing and case preparation

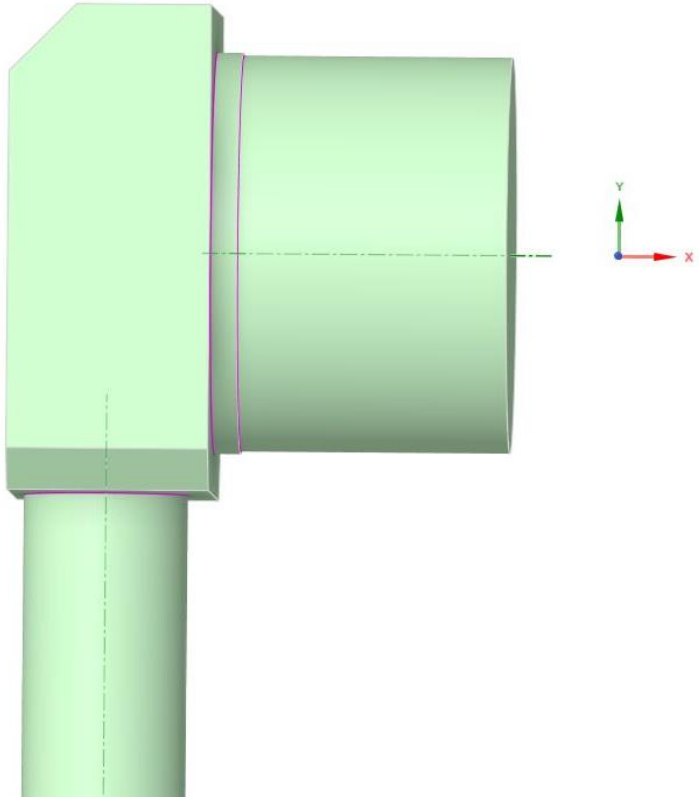


Tosca fluid optimisation

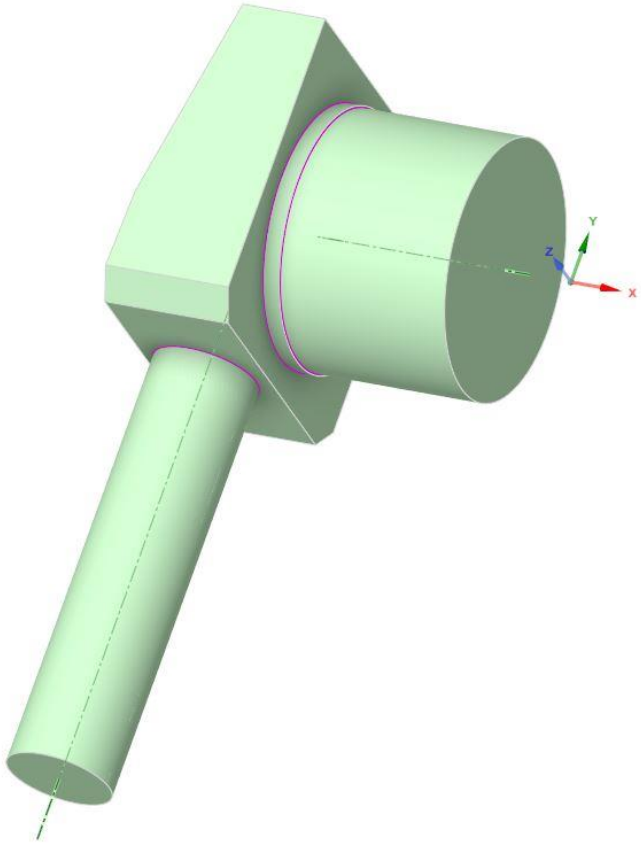
Preparing model in spaceclaim for fluent



Model 1: Large area for recirculation



Model 2: Reduced area pushing outward (left)



Model 3: Draft on the back wall aimed to redirect flow

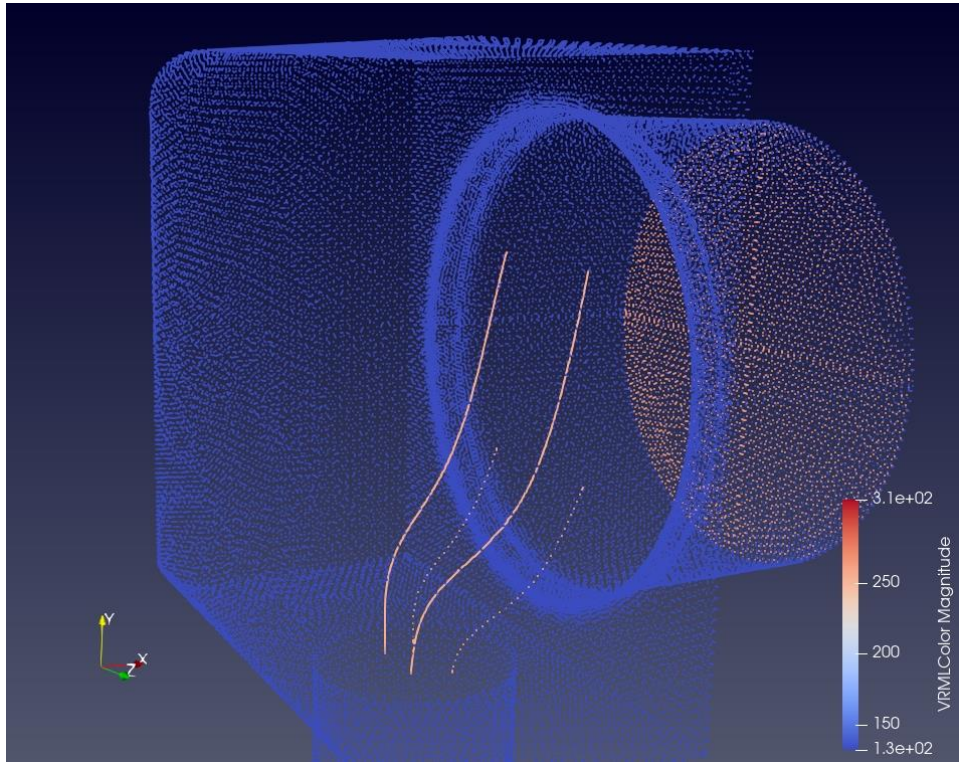
Insights from various cases

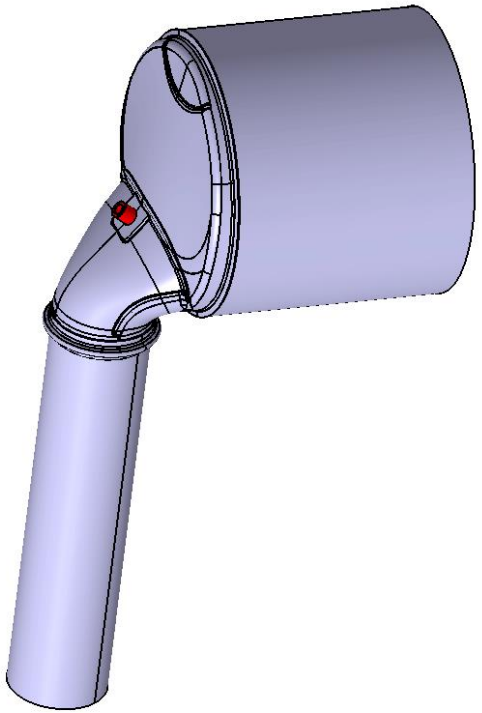
- Always import the model to spaceclaim when using Ansys
- This allows us to check for errors in the model and apply 'share topology'
- 'Share topology' eliminates the need for creating mesh interfaces
- Not checking the model in SpaceClaim resulted in issues in Tosca fluid

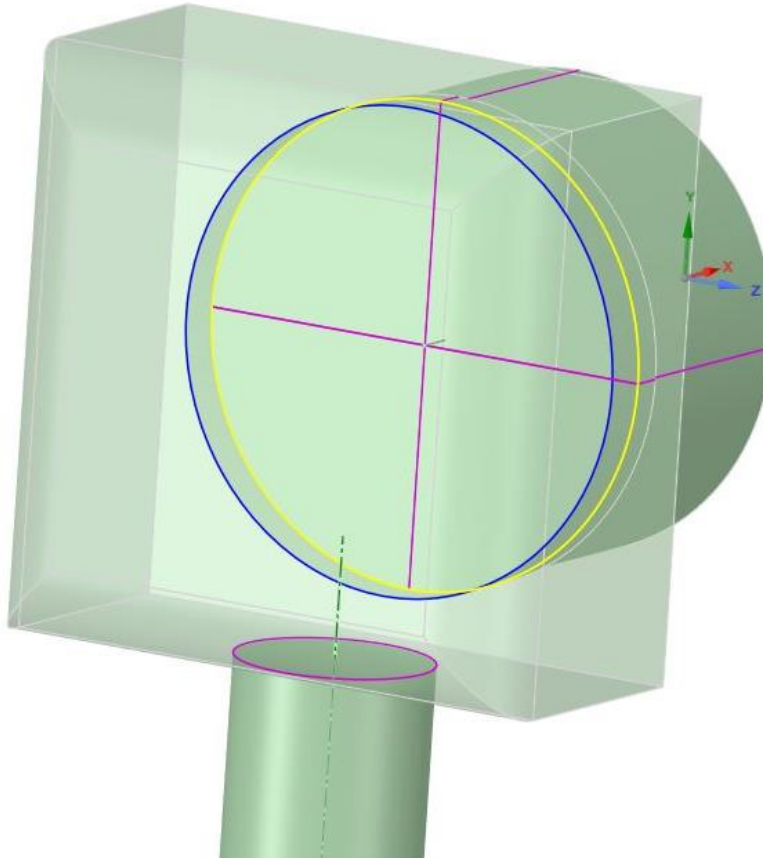
How did the particle tracks turn out ?

Outlet as 4 quadrants:

- Divide the outlet of design space (outlet interface) into 4 quadrants

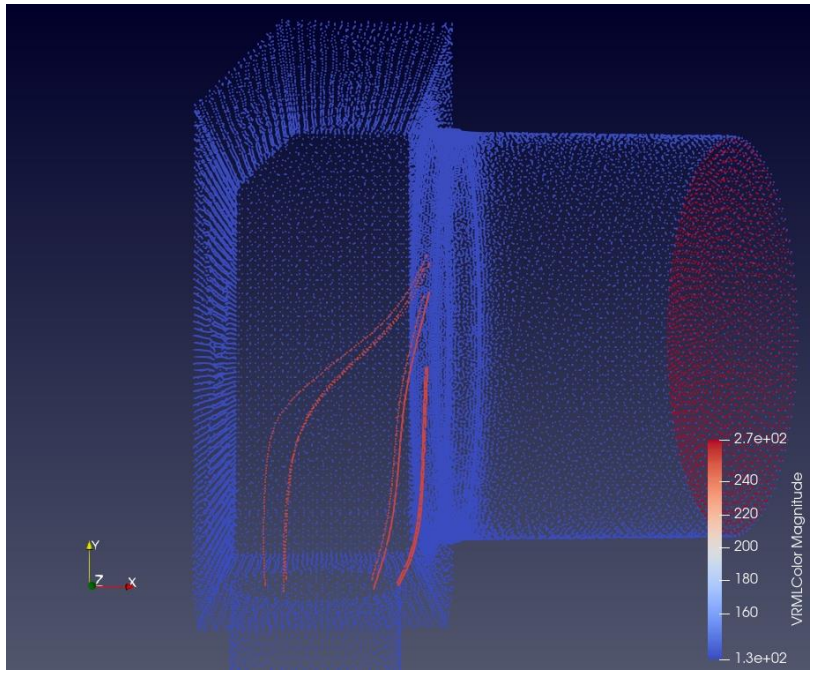


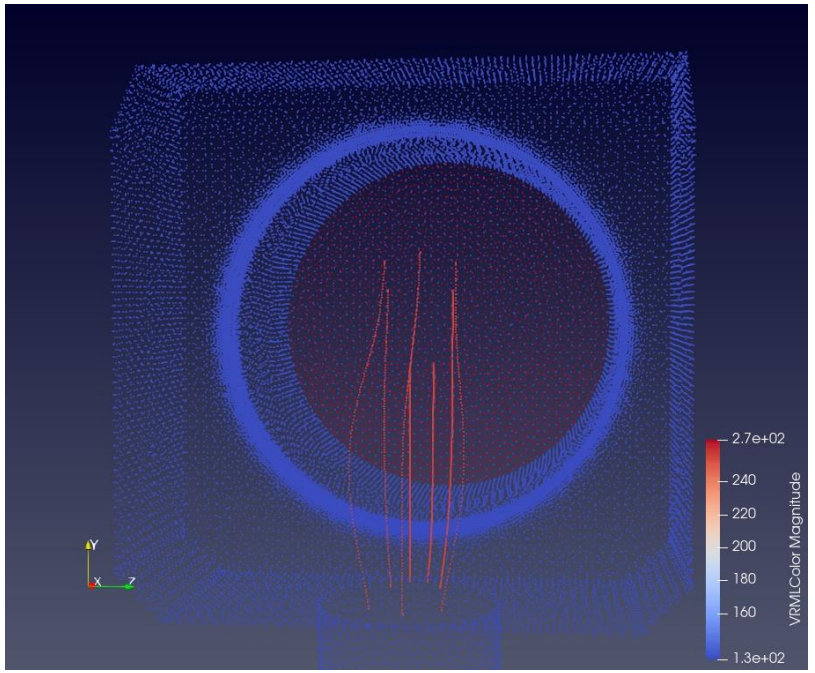


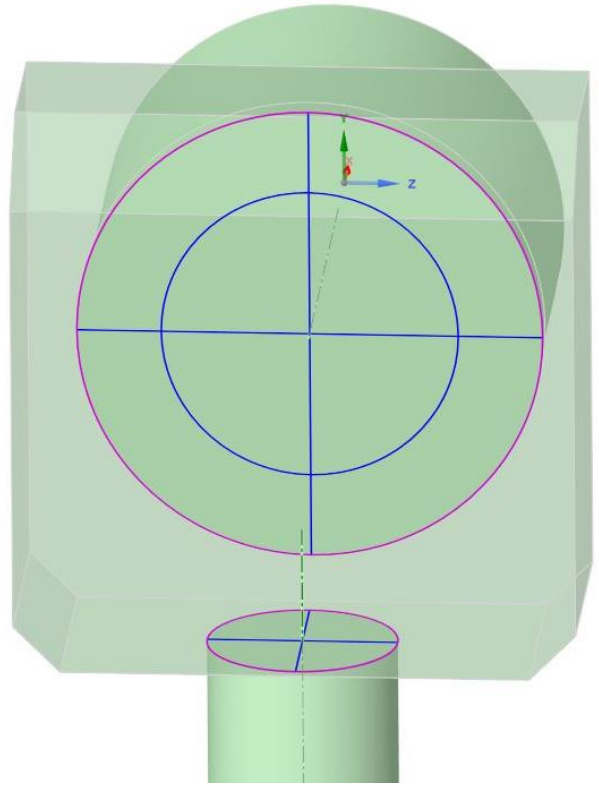


Outlet as 8 divisions & inlet as 4 divisions:

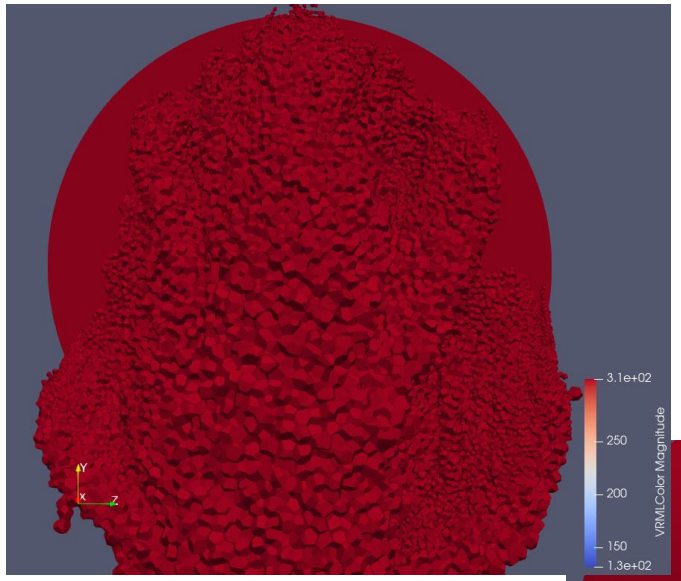
- More inlet outlet does not mean more particle tracks

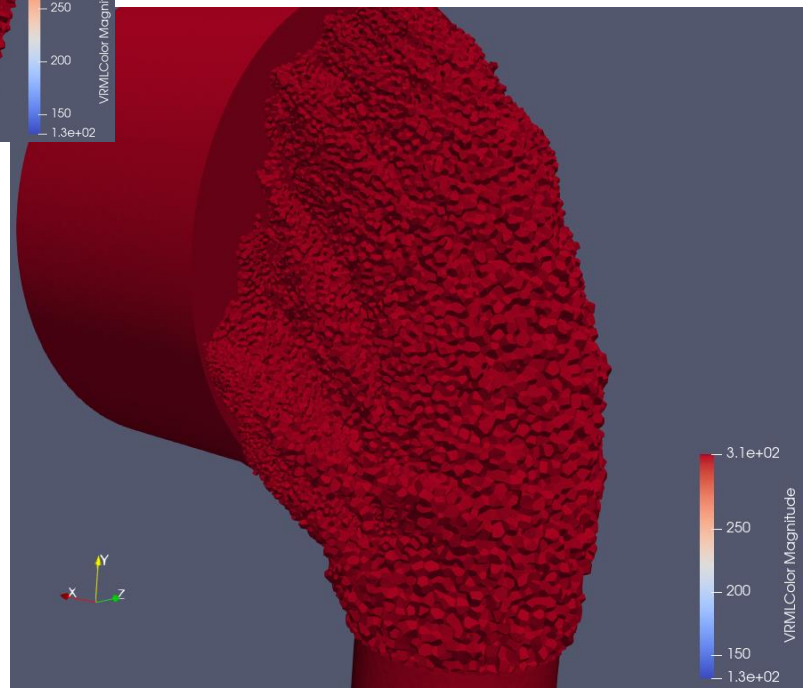


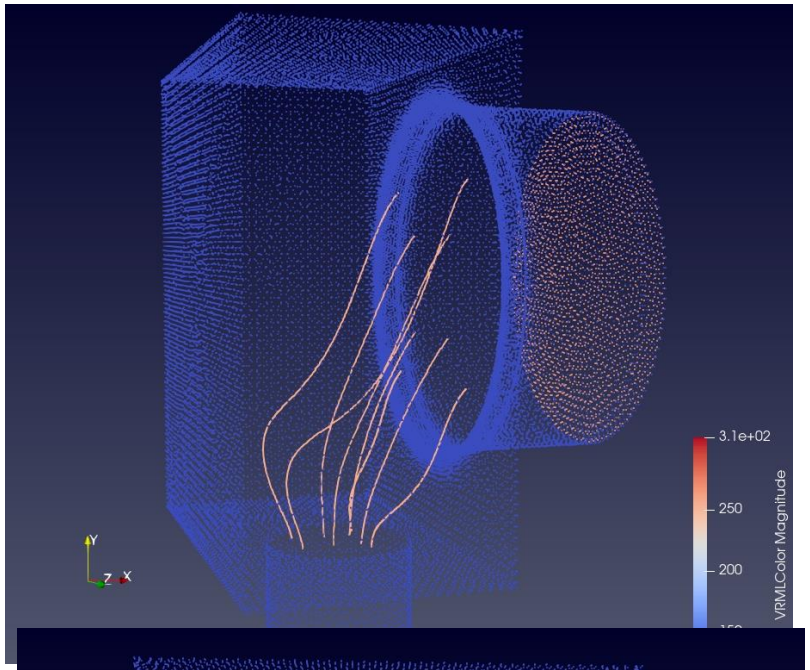


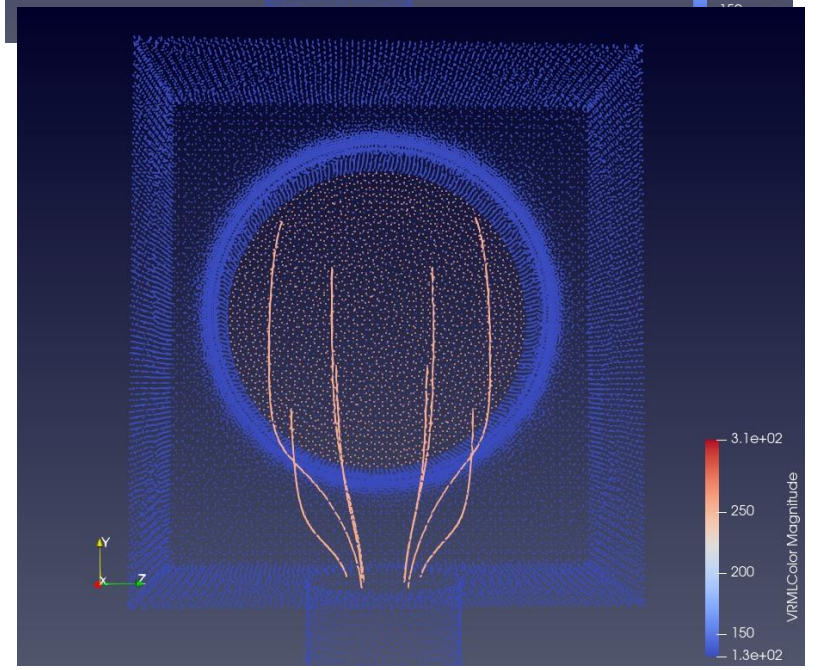


- Endplate (current model)



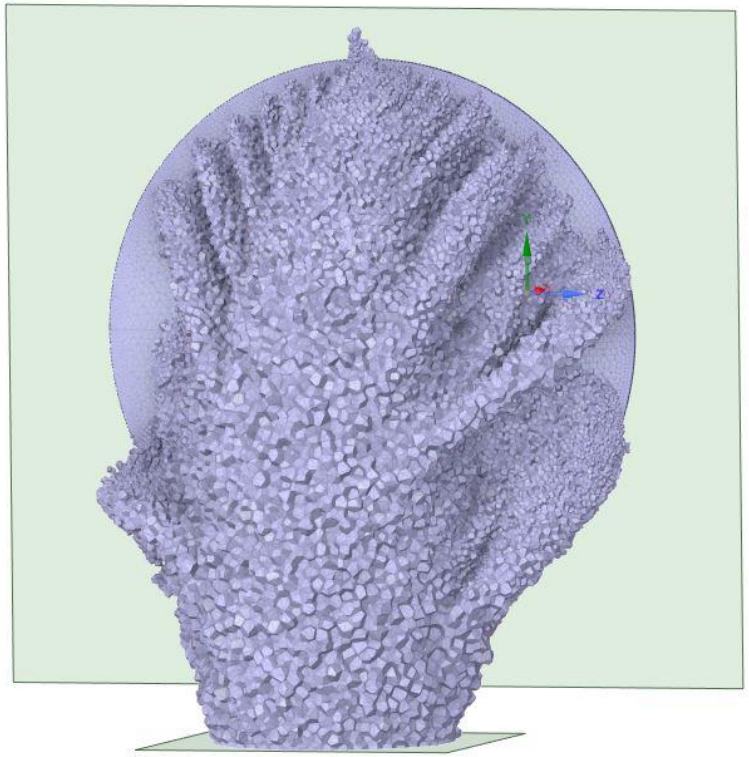


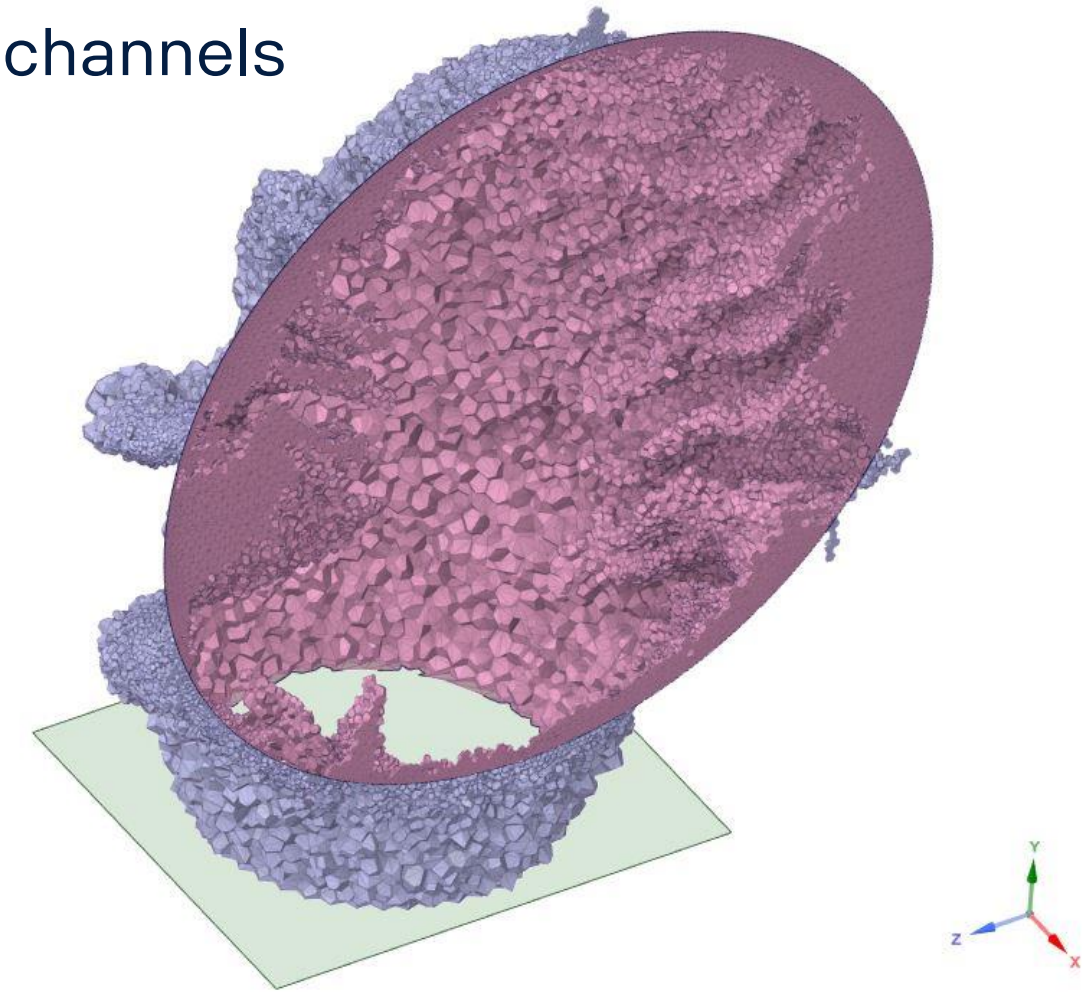




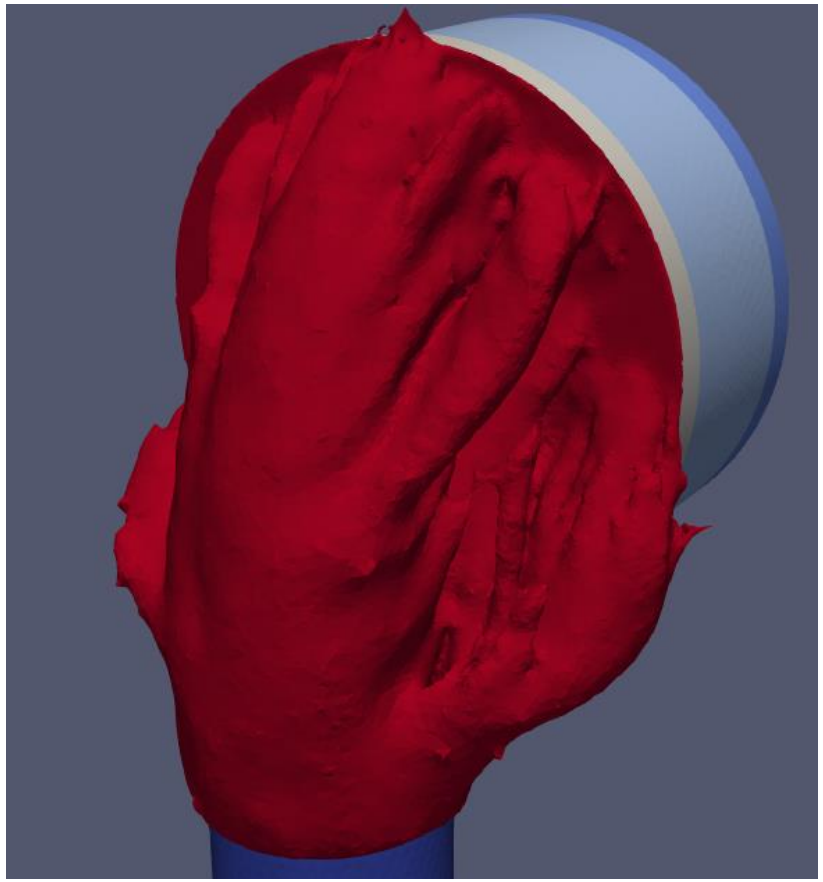
Importing the generated design to spaceclaim

Interior region with material forming channels

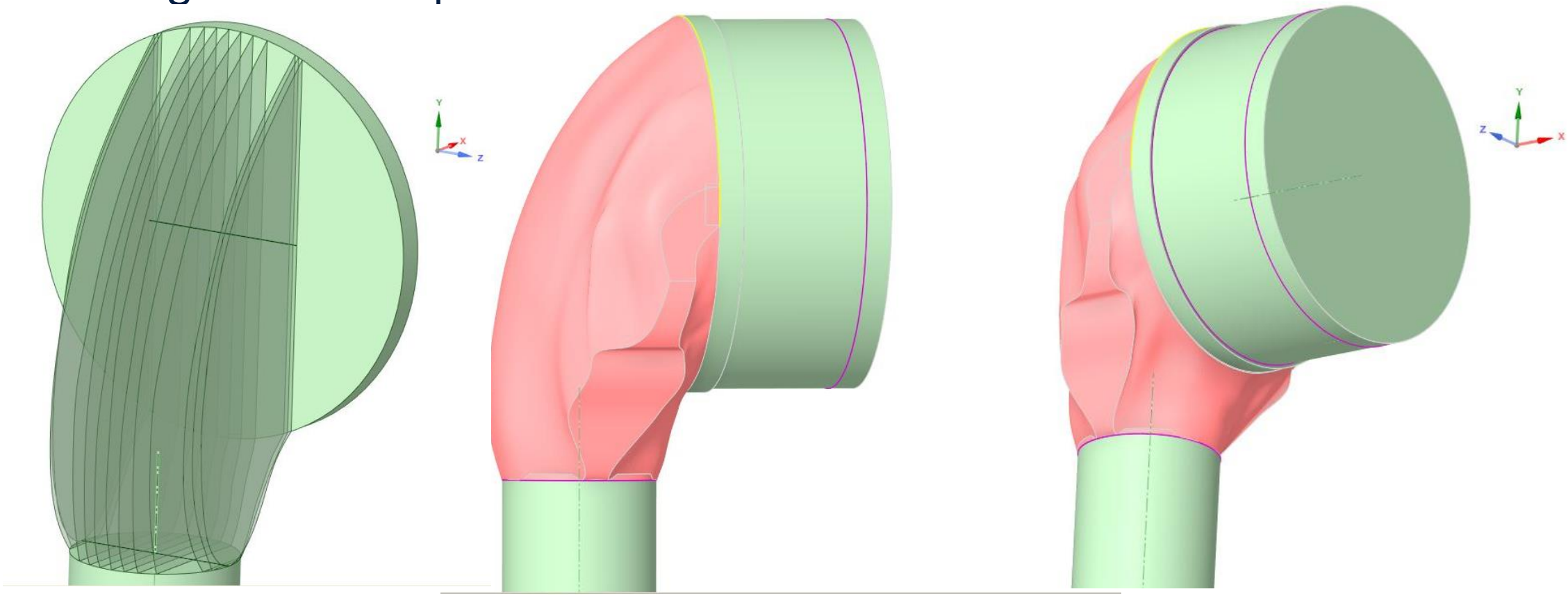




Smoothening done on Tosca Fluid smooth



- Alternate method to smoothen and recreate the resulting geometry – Ansys space claim:
- 1. slicing and blending surfaces along z-axis
- 2. using shrink wrap feature



Insights

- Test the fluent case with laminar flow to get the idea of how particle track would look like – SAVES TIME running the job on cluster
- Dividing inlet and outlet of DS does not necessarily mean particles tracks are generated between them
- Design with better particle tracks (better spread across the DS outlet interface) results in better UI
- TOSCA FLUID optimises only backflow, this makes it difficult to use for optimising for better UI across substrate

Upcoming work

- For research worthiness/ academic part of thesis: Evaluate what variations in the model can bring about better particle tracks and hence better solutions.
- Create a model study (with a parametric model) to find how (say changes in) chamfer at a corner affects ui, pressure drop and backflow when using tosca fluid
- Determine if tosca can be useful for applications involving substrate (Here UI is the determining criteria)