§

Optimization of XC40 cabinHeatup Results in EVs at Low Temperature

Hussain Abbas

Supervised by

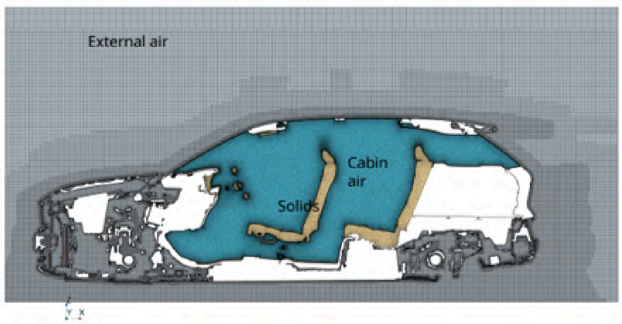
Randi Franzke
Emil Wileson

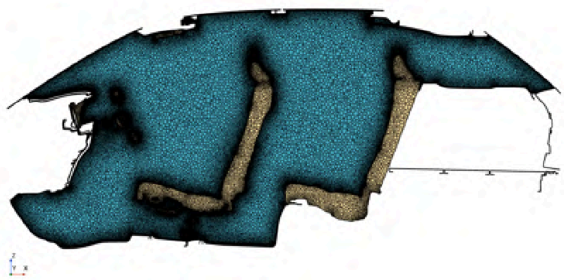
Table of Contents

Table of Figures

Introduction.

The project is continuation of what Anandh, a PhD student did couple of years ago. His study was performed at a speed of 50km/hr and a temperature of -4C. At 920 cores his simulation took 115hrs to finish and the average temperature within the cabin was 24,7 C. My principal tasks involved optimization of his results without compromising the accuracy of model. I have to find a method with which we could reduce the simulation time of 115hrs to less than 24 hrs without deviating to more than 1C (10%). An easy way would have been to coarsen the mesh as it would reduce the computational time but we didn't wanted our mesh to get too coarse. The whole study was divided into different parts so the effect of each individual parameter can be analyzed accurately.





Anandh PhD Project
XC40 CabinHeatup at low temp

Initial Conditions
- Velocity 50km/hr
- Exterior Temperature -7C
- Cabin Initial Temp -4C
- Temp HVAC 30C
- Heating Time 40mins

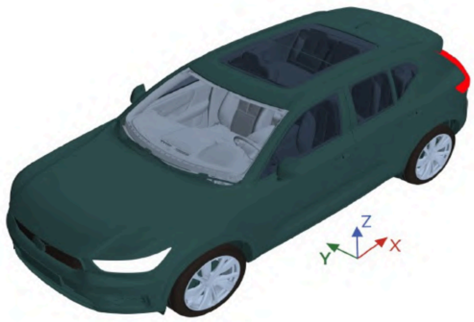
Final Results:
Ncpus 920
Avg Cabin Air 24.7C

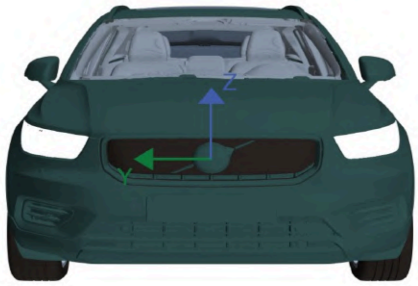
Compt Time 115hr /4.8days

Here is the sequence of steps in which the whole study was carried on.

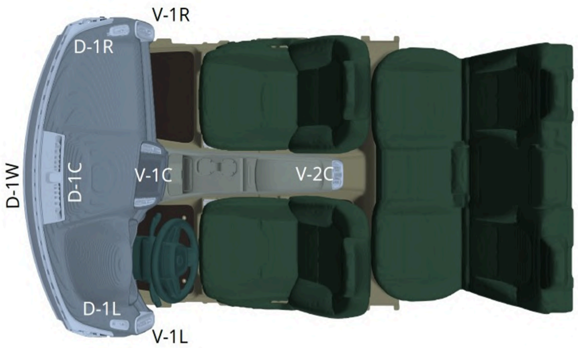
1. Geometry cleanup
2. Effect of Timestep on computational time
3. Effect of Sub-iterations on computational time
4. Effect of Prism layers on computational time
5. Effect of changing mesh settings (minimum surface/target surface)

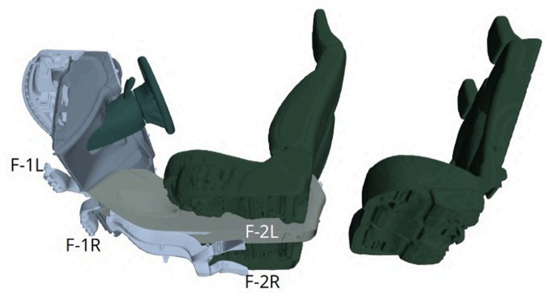
Vehicle used : XC – 40 BEV.





D-1R : Def, 1st row, right
V-2C: Vent, 2nd row, center
F-1L: Floor, 1st row, left





CFD modeling: Computational domains

Three computational domains:

Cabin air (CA)

Ideal gas (with const density init.)
Unsteady (with steady init.),
Segregated flow and temperature,
Realizable k-ε turbulence model,
Gravity

Cabin solids (CS)

Multi-part solid,
Unsteady,
Segregated solid energy

External air (EA)

Constant density,
Steady,
Segregated flow and temperature
SST k-ω turbulence model

Implicit heat transfer coupling

Explicit heat transfer coupling

Improvements.

Three improvements were done before proceeding with the simulation.

Using large Monitor volumes instead of small volumes.

For examining the distribution of temperature all over the cabin, Anandh used 9 monitor volumes of size (10 by 10 cm) at different locations (3 for driver side, 3 for passenger side and 3 for rear side). The problem with those small volumes was that they were too small to accurately calculate the distribution of temperature at those positions. Second, a small change in volume will change the direction of jet and might miss the small volumes. So instead of using small volumes, large monitor volumes of around (40 by 40 cm) are used to accurately calculate the distribution of temperature.

Figure 1: Small and larger monitor volumes for calculation of temperatures at those areas

Using mean of last N-samples Temperature reports instead of instantaneous value reports.

Another important evaluation technique that differ from Anandh analysis was taking mean of last n-samples (iterations or timesteps). The reason was that in plots of individual monitors there was large deviations (peaks). Because of that we decided to use mean reports instead of instantaneous temperature reports. We used 50 secs mean (2500 timesteps) as reference.

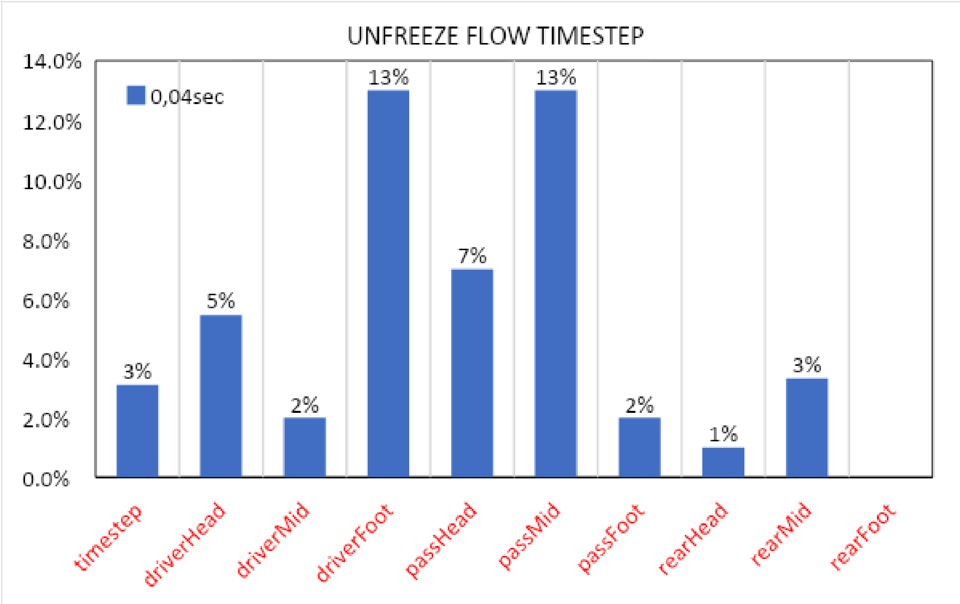


Figure 2: Using instantaneous temperature reports for monitor volumes

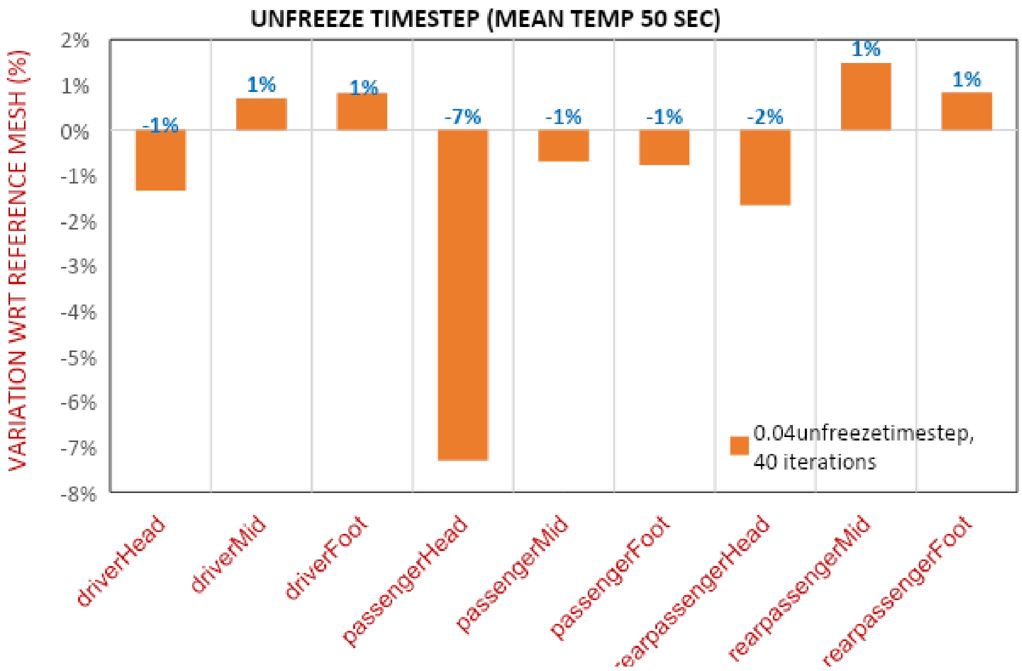


Figure 2: Using mean (50secs) temperature reports for monitor volumes

In the above plots, we can see that when we were using instantaneous temperature reports, our deviation was more but when we move to instantaneous reports, the deviation decreased.

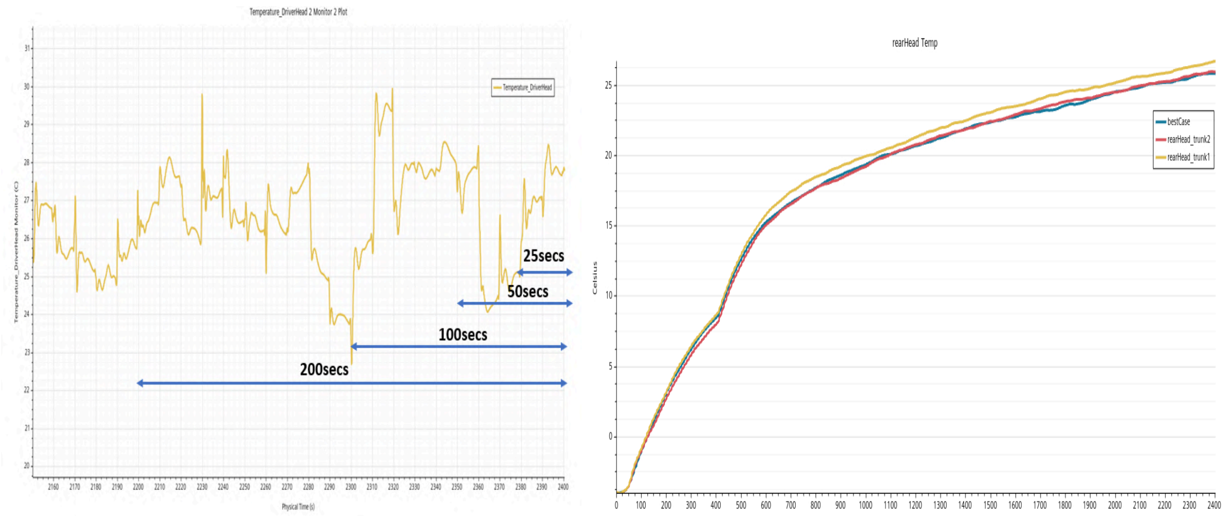


Figure 2: Left (instantaneous temperature report) plot vs Right (50 sec mean) temperature report

Evaluation method

In the report that Anandh published, monitor volumes temperatures has been calculated as normalized mean temperature so in order to know how much our value deviate from anandh case, normalized mean temperature of our model is subtracted from normalize mean temp of Anandh model.

Normalised Mean Temp of Monitor

where

Thvac= 30C, Ti= - 4C

Variation wrt Finemesh/timestep =

,

Reference Mesh :

Anandh Mesh at

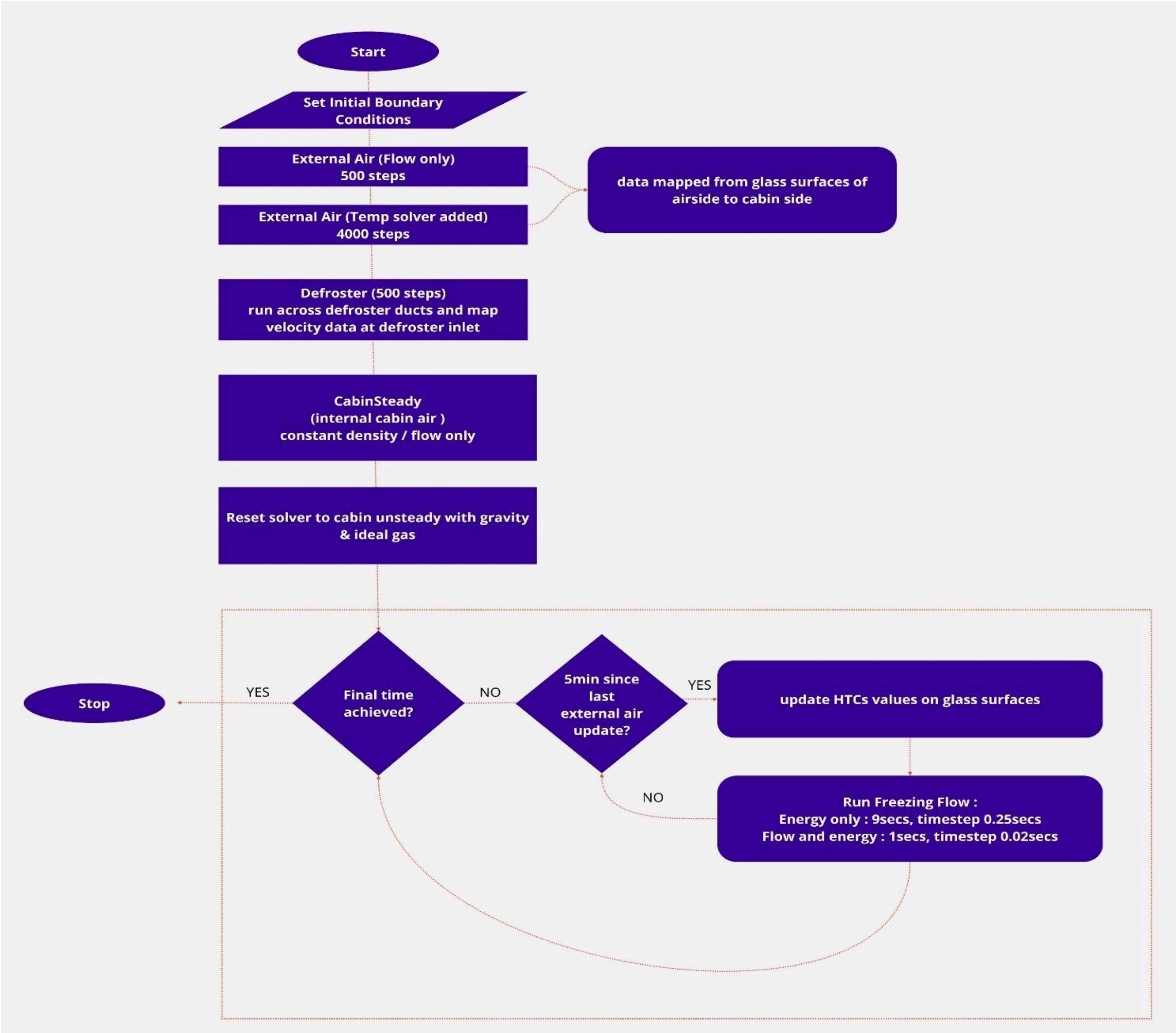
0.01sec unfreeze timestep,

0.25sec freeze timestep

Here first normalized mean temperature for each monitor is calculated. Tmonitor is temperature of individual monitor, Ti in is the initial temperature (-4C/268.15K) while Tinlet is the max inlet temperature to the HVAC (30C /303K). Both these values are takes from the settings used by Anandh.

Then based on the above formula, deltas for each monitor is being calculated.

Simulation Framework



Initialization – The procedure start with initialization of initial and boundary condition such as initial temperatures of domain, vehicle velocity, mass flow rate and temperature of air at cabin inlets and final physical time for the cabin heatup.

Running Aero – Simulation is run on the outside of vehicle only for flow with no temperature data. It uses the script AeroFlow. The simulation is run for 500 steps (1st order) then for 1000 steps (2nd order).

Running Airstream – Then for 4000 steps, simulation is being run with temperature solver added. At the end, temperature data is being mapped from solids/airstream. CHT interfaces are setup between solids in contact with external air (front/rear windshield, sunroof glass). At the end of this step, HTC data computed on external boundaries is being mapped as convection boundary condition for solids (from solids/airstream (front/rear windshield, sunroof glass) to solids/cabinair.

Run Defroster Flow- Simulation is being run in defroster ducts for 500 steps and at the end, velocity data is mapped at the inlet of defroster.

Run cabin steady flow- Simulation is being run in cabin for 5500 steps. Only flow solver in the cabin to give us a rough estimate of flow within a cabin. This will serve as a starting point for unsteady flow in cabin. The internal volume of air in the cabin is modelled as a gas with constant density during domain initialization. During heating, internal volume is modelled as an ideal gas. In order to save computational time, the solver is set to steady state first and then switched to unsteady. Done only for cabin air without solids

Gravity- its added to the solver before cabin unsteady flow to account for the effects of buoyancy during heating while the effects of solar radiation is neglected because of their low impact in cold weather.

Run Cabin Unsteady flow- The solver is switched to unsteady flow for both cabin air and solids, with a timestep of 0.02 secs. Once condition of steady state is being reached at monitor volumes for velocity a loop is being made.

First condition – is a time based stopping criteria to terminate the simulation if physical time is being reached (2400 secs)

Second condition- is to recompute the external flow and HTC on external interfaces every five minutes. In the meantime we freeze the flow for 9 seconds with a timestep of 0.25sec and then unfreeze flow for a 1 seconds with a timestep of 0.02secs. After 5 mins has reached we update the value of HTC on external interfaces if there is any change and the loop continues until we achieve a final physical time of 2400secs.

Geometry Cleanup

During my initial meetings with Anandh, he talked a lot about improvements needed to be done in the geometry. Considering that I analysed some of important section where we had high gradients such as vents and front area of cabin. I found that in majority of the boundaries, the prism layers were not available to accurately capture the boundary layers. All the geometry cleanup was done in Ansa and Starccm+ Surface Repair tool. Because of those minor changes in the geometry, the new simulation was named CORRECTED MESH.

Everywhere you see something by the name cabinHeatup, it refers to Anandh original file on which he worked and which was my starting case while corrected Mesh refers to the improved mesh that I obtained after cleanup. Anandh mesh had 33million cells in solids and 100 million cells in cabin while corrMesh has 33 million cells in solids and 95 million cells in cabin. Below I have attached a picture which shows comparison of anandhMesh prism layers and my corrMesh model prism layer. Boundary layers are captured good in corrMesh compared to Anandh cabinHeatup model.

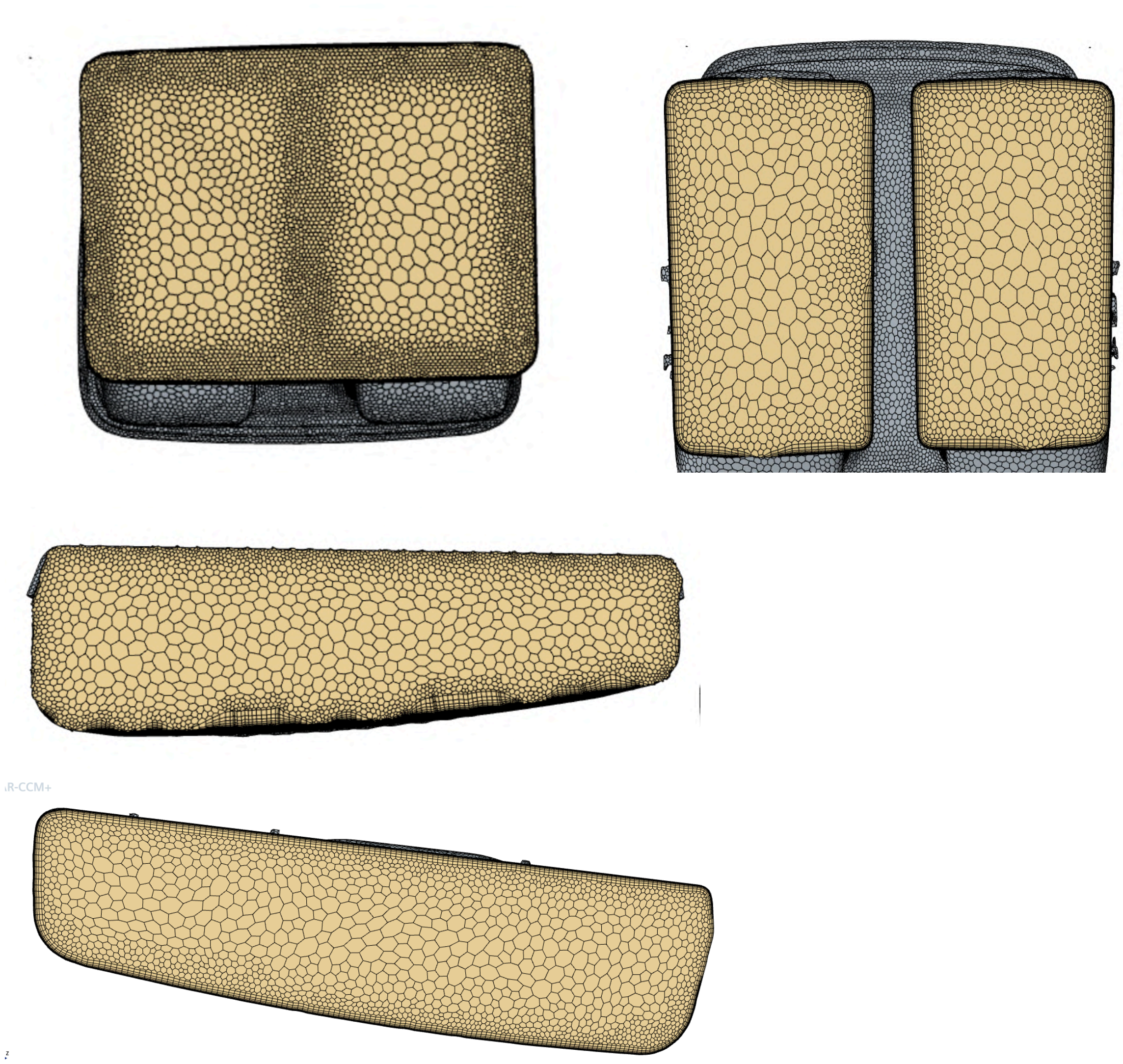


Figure 3: Improvement of Prism layers around vents in Anandh starting case and corrMesh

cabinCells	
cabinHeatup Mesh	100
corrMesh	95

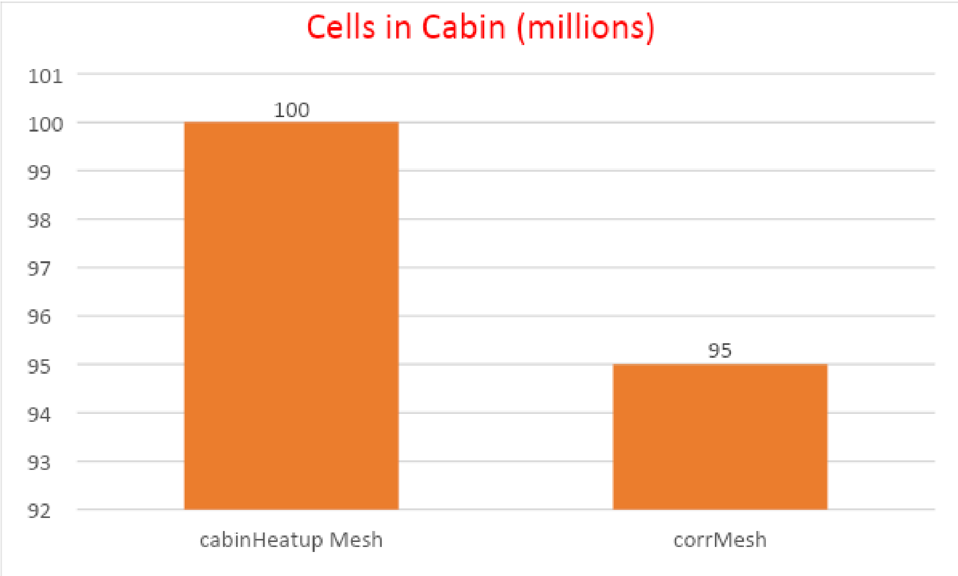


Figure 4: Total no of cabin cells in anandhStartingCase and correctedMesh

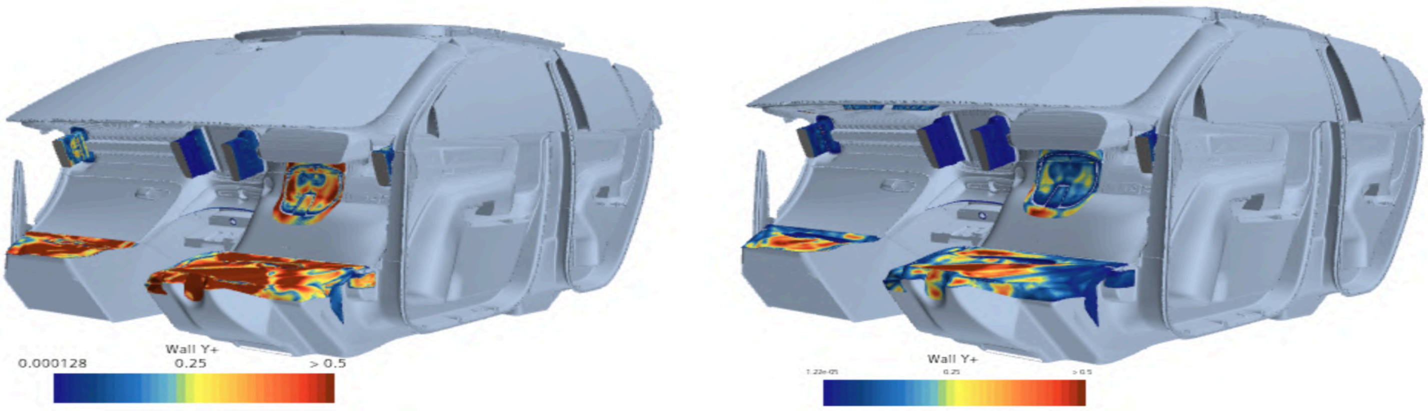


Figure 4: Wall-Y+ comparison (left) anandhStartingCase and (right) correctedMesh

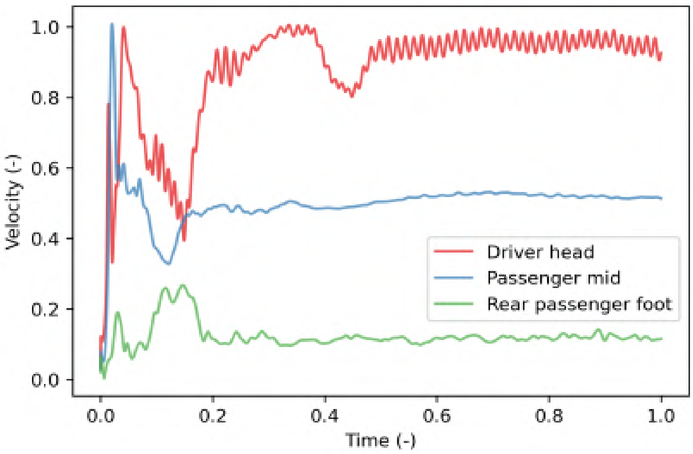
If we compare the Y+ wall between Anandh starting case and corrected mesh, we see quite some improvement around boundaries. Majority of them have a wall y+ less than 0.5 which meets our desired set criteria.

Timestep study.

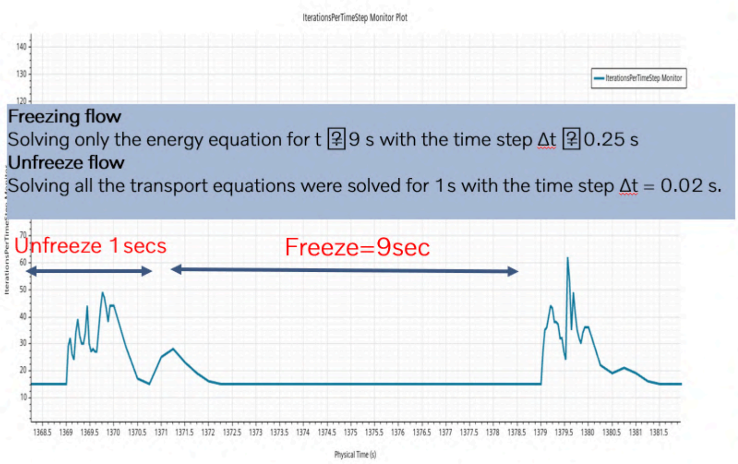
Timestep is another important parameter that I analysed in reducing solver computational time time.

In cabinHeatup simulations, we use both steady and transient simulations. First we run steady state simulation of internal air with constant density to reduce computational effort. Since after turning on the blowers, the velocity reaches steady state within few seconds. Then we switch the solver to transient unsteady with ideal gas and gravity.

In transient simulations, Anandh did comparative analysis how changing timestep can effect solver elapsed time. He used timestep during unsteady transient simulation. The major reason was that the velocity within the cabin after some time becomes steady so in such case, timestep can save you a lot of computational time.

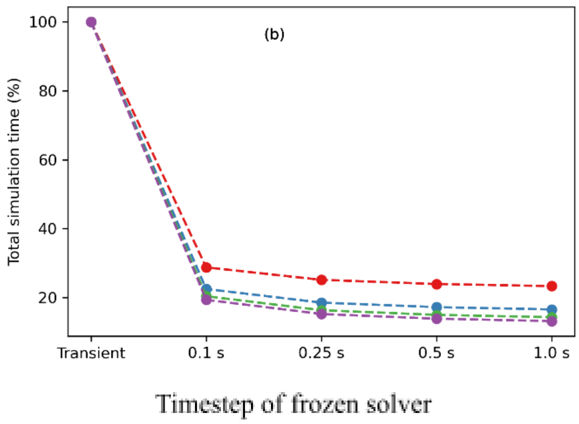


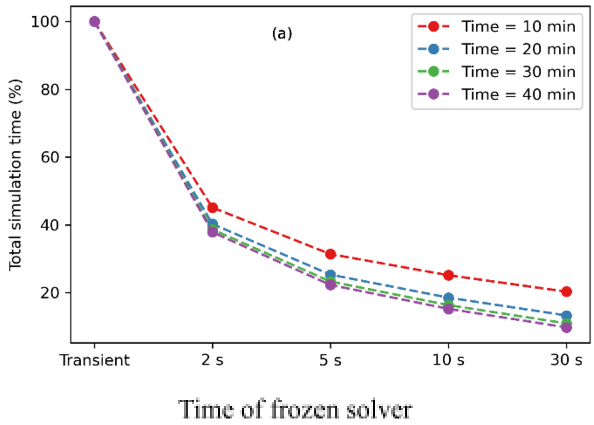
Inorder to speed up the process, the flow was freezed for 9 seconds in which only energy equations were solved. Here the timestep used was 0,25secs. Then the flow was unfreezed for 1 seconds in which all the transport equations (flow and energy) were solved to support convergence. Here the timestep used was 0,02 secs. Anandh did a comparative analysis on the effect of both freezing and unfreezing timestep on total simulation time. His results show that for freezing flow, increasing the timestep (from 0,25 seconds to 0,5 seconds) and freezing time (from 9 secs to 30 seconds) has a very small effect on decreasing simulation time. That's the reason that for our study, we kept the settings for freezing flow (timestep and time) same as what was used by Anandh.



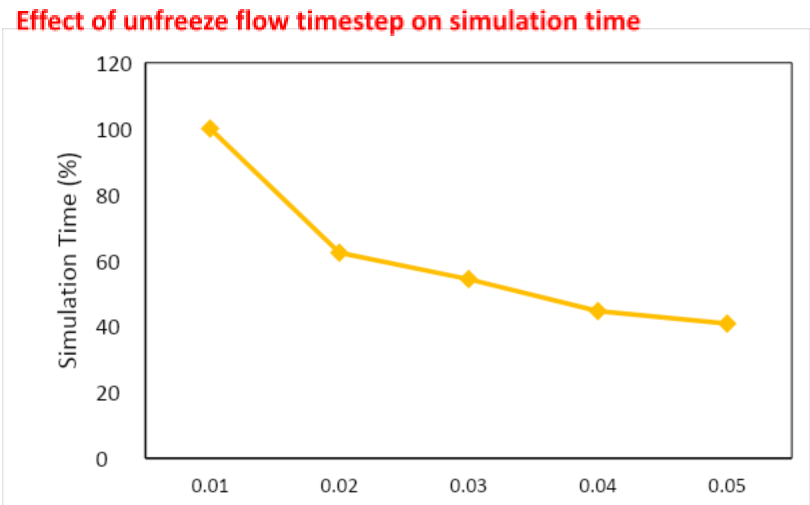

```
// Simulation settings:
double defrosterSteps = 500;
double airStreamSteps = 4000;
double cabSteadySteps = 5000;
double totalTime = 2400.0;
double tsUnsteady = 10.0;
double freezeTime = 9.0;
double unfreezeTime = 1.0;
double freezeTimeStep = 0.25;
double unfreezeTimeStep = 0.02;
String solverType = "Second";
String saveFile = "java"; //"sim" or "java"
boolean gravity = true;
boolean simHistory = true;
boolean srh =true;
    // Boundary & Initial conditions:
double U_ref = 50/3.6; //kph
double T_ref = 266.92; //K
double cabInitialTemp = 268.55; //K
double solidsInitialTemp = 267.15; //K
double tempInlet_HVAC = 303.15; //K
```

Figure 5: Settings used for freezing / unfreezing flow during the transient simulations



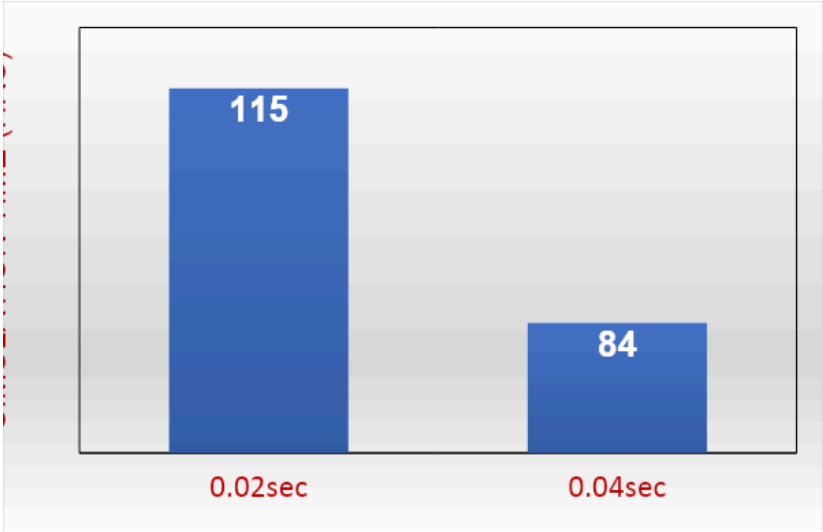


Now the point is that which parameter will help us reduce our timestep so using a larger timestep could provide some improvement. That's why I plotted simulation time against both freezing flow timestep (0.25sec) and unfreezing flow timestep(0.02sec). In the frozen time, we conclude that using larger timestep(0.5 instead 0f 0.25) wont improve the simulation time significantly so we kept it same as what Anandh used.



But plotting the simulation time against unfreeze flow timestep, we see that using a larger timestep will help us reduce the computational time. So instead of using 0,02sec which Anandh used, we use 0,04secs.

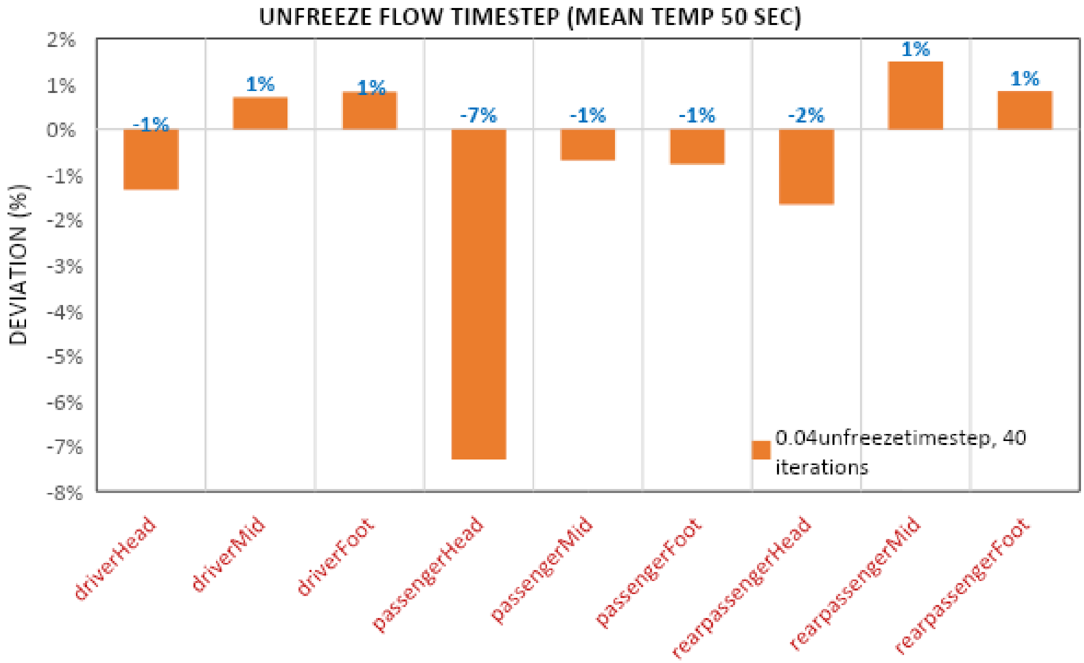
Reduction in Simulation time due to using large timestep (0.02 ->0.04)



After using larger timestep for unfreeze flow (0.02>0.04) we saw a decrease in simulation time by almost 30 hrs. Our simulation time was decreased from 115hrs to 84hrs.

But this improvement comes at a cost of deviation in temperature, deviation wrt to Anandh reference case shows some deviation but its within our desired criteria. (less than 1C)

Deviation in Normalised Mean Temperature at 50 secs wrt ReferenceMesh at 0,04sec



Majority of the monitor volumes show a deviation of less than 0.5C except passenger head where we have a higher deviation. This might be because on driver position the heat reaches the head position through steering so some heat is lost in those position but in passenger case, we don’t have such case.

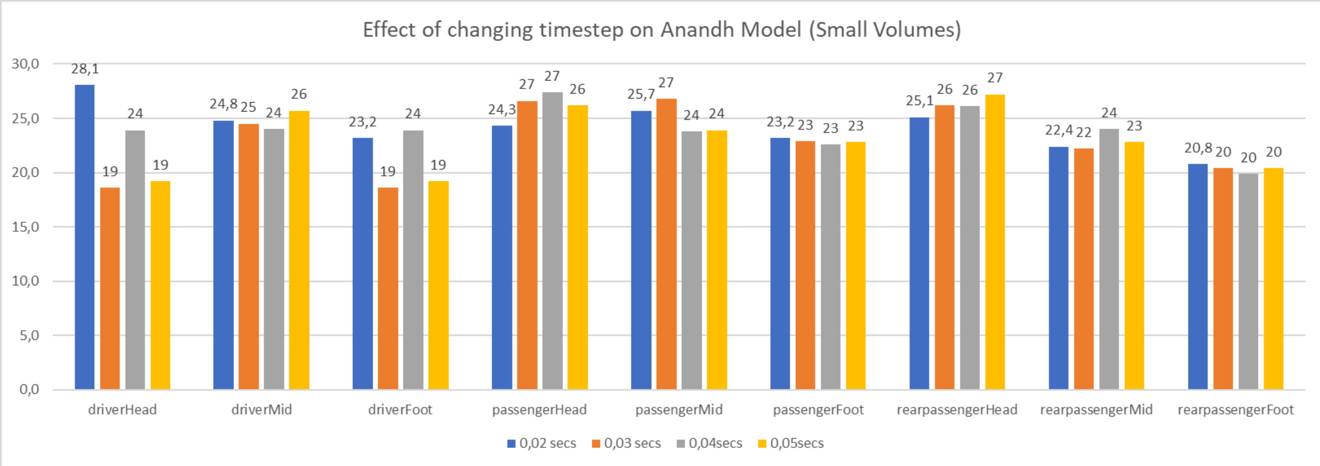


Table 1: Effect of changing unfreeze timestep on temperature of small monitor volumes of Anandh Starting case.

Comparison of instantaneous temperature reports and small volumes shows that for most of the monitors the deviation from base settings (0,02secs) deviation more than 1C. (Important thing to note that this is data of small monitor volumes which is also one of the reason we are seeing deviation of 1C. When using large volumes, youll see a reduction in this deviation from base setting decrease to less than 0,5 secs.Below is the bar plot for large volumes (Anandh Model)

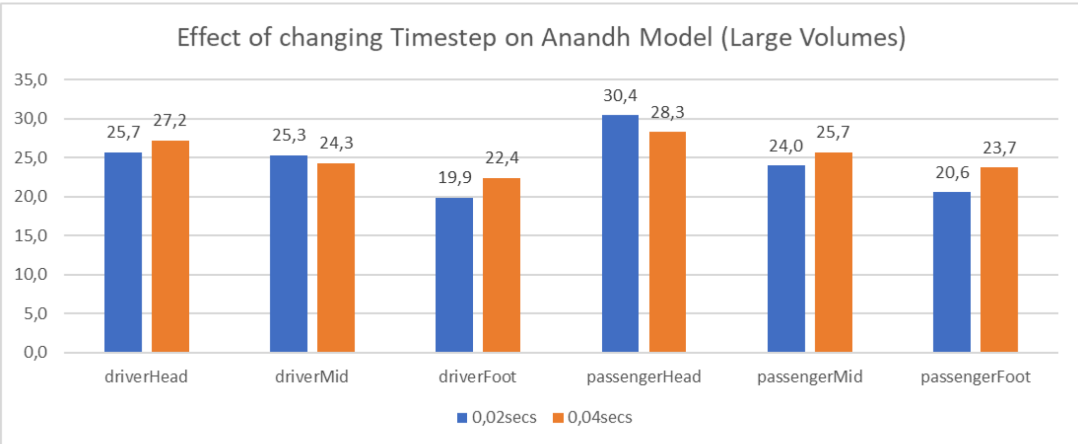
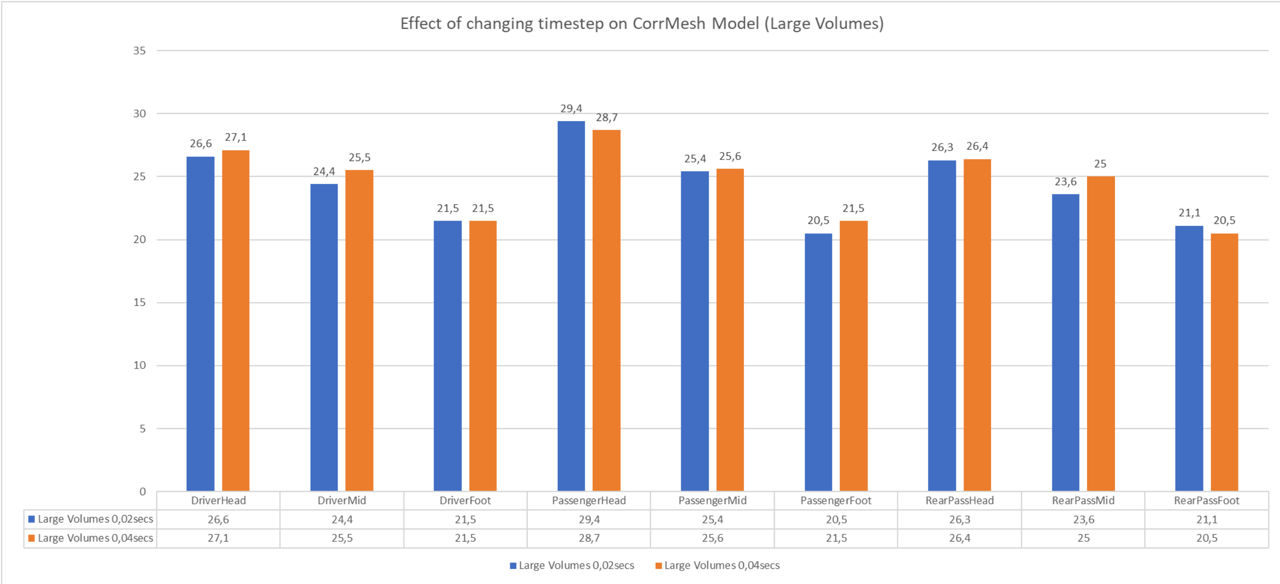


Table 2: Effect of changing unfreeze timestep on temperature of large monitor volumes of Anandh Starting case.

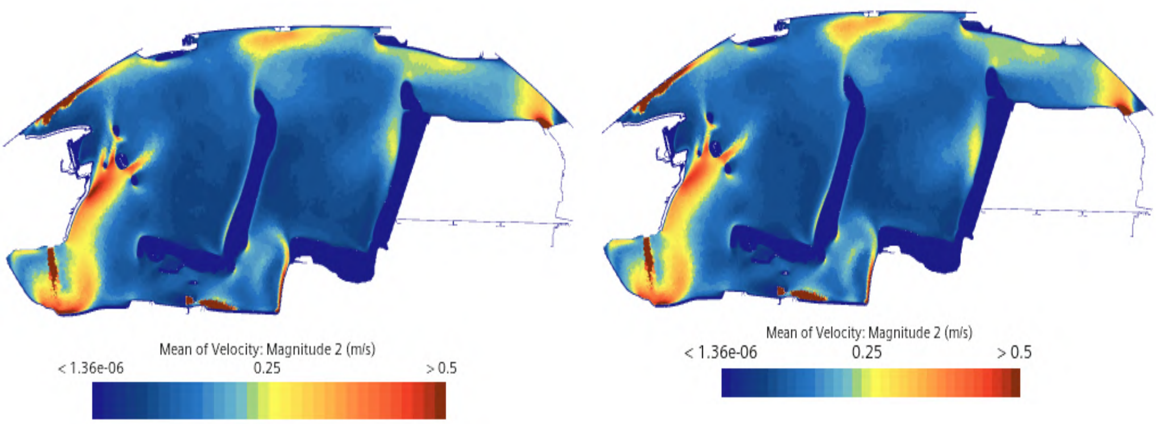
But all the above graphs were for anandh model, we want to see the effect of timestep on our model that we got after doing the geometry cleanup (corrMesh model)



	DriverHead	DriverMid	DriverFoot	PassengerHead	PassengerMid	PassengerFoot	RearPassHead	RearPassMid	RearPassFoot
Large Volumes 0,02secs	26,6	24,4	21,5	29,4	25,4	20,5	26,3	23,6	21,1
Large Volumes 0,04secs	27,1	25,5	21,5	28,7	25,6	21,5	26,4	25	20,5

Table 3: Effect of changing timestep on temperature of large monitor volumes of improved mesh (CorrMesh).

Results from timestep study was that by coarsening the timestep for unfreezing flow, (from 0,02sec to 0,04secs), we saw a reduction of almost 30 hours while the temperature deviation in majority of monitors (large monitors) was below 0,5C which is a good start . Comparison of both anandhmesh and corrMesh are shown below against changing timestep.

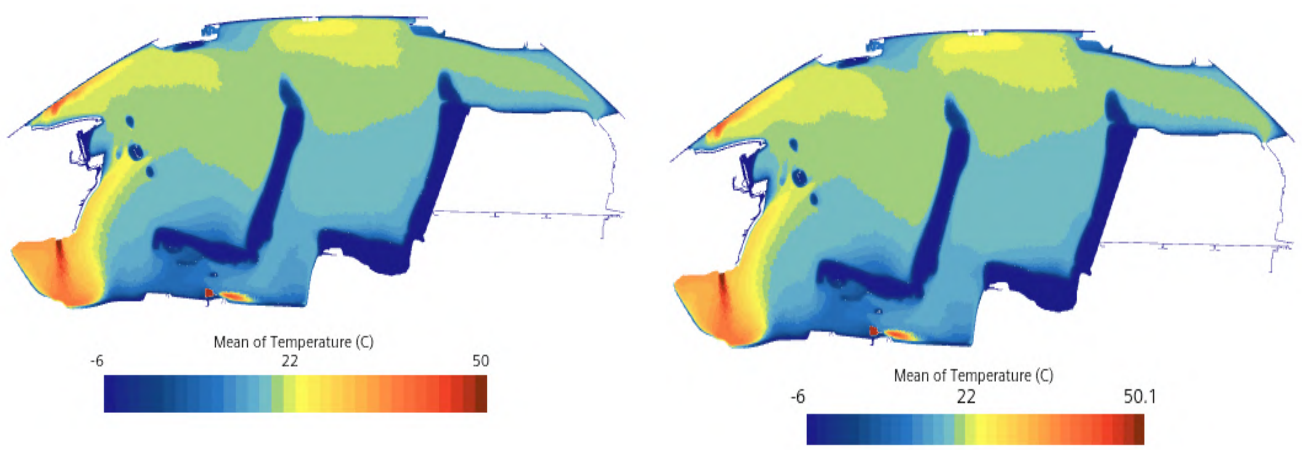


Comparison of Mean Velocity (AnandhBaseCase vs afterTimestepStudy)

Looking at the velocity profile shows a very small change. At the inlets, windshield we see a similar profile of velocity.

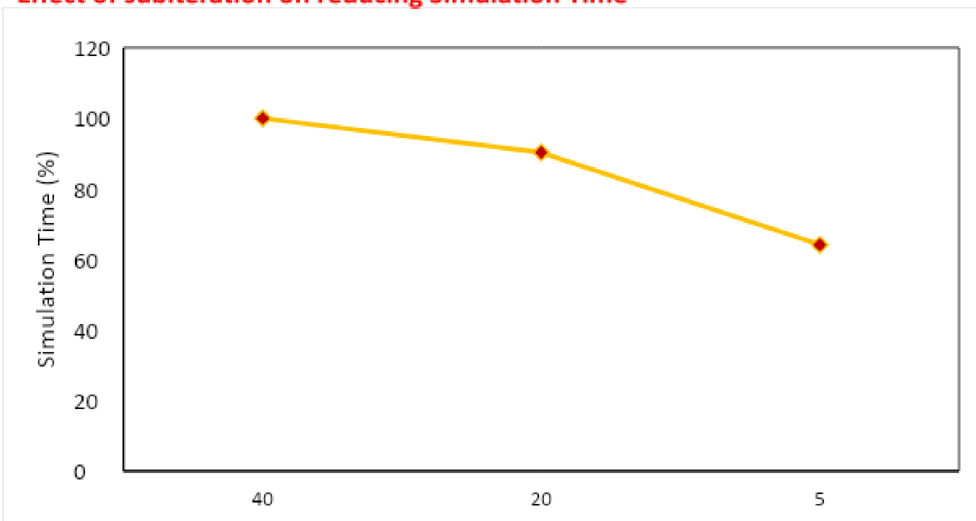
Same goes for the temperature profile, except the outlet, everywhere the heat distribution is same.

Comparison of Mean Temperature (AnandhBaseCase vs afterTimestepStudy)



Sub-Iterations Study

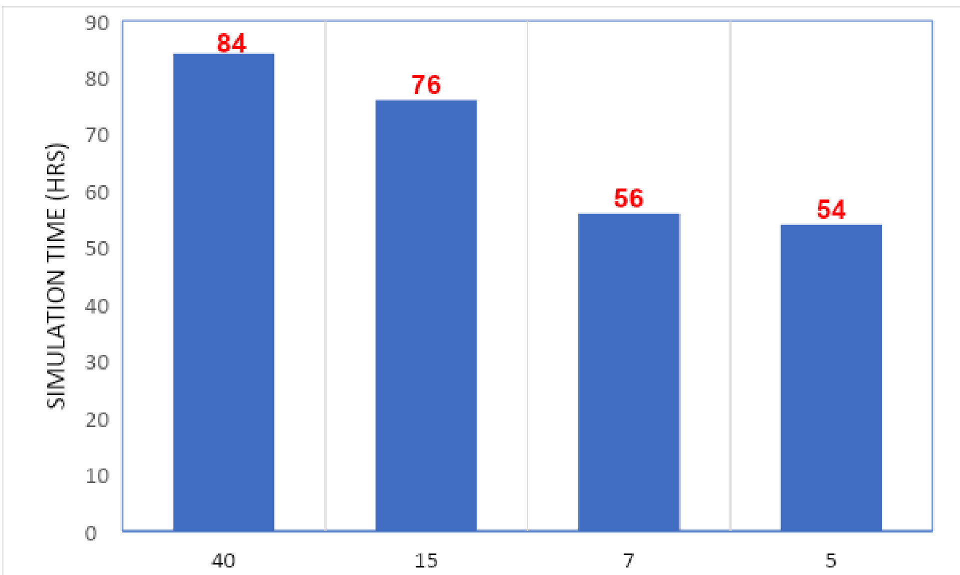
Effect of subiteration on reducing Simulation Time



Second thing to analyze was number of minimum iterations used within one timestep. Anandh was using 40 iterations in his base case which is too high. In order to do the study, first comparative analysis the accurate number of iterations was necessary. In order to know how many iterations are sufficient to reach convergence, spherical point were created around high, intermediate and low velocity areas in the cabin. Temperature reports for those points were created to check beyond how much iterations we don't see any change between two successive iterations. Results were that after 5-7 iterations, we don't see any change between consecutive iterations. But in order to capture the overall effect, four case studies are done (40,15, 7, 5 iterations).

By doing so, we further decrease the simulation time by almost 31 hrs.

Reduction in Simulation time due to decreasing sub-iterations (40 -> 5)



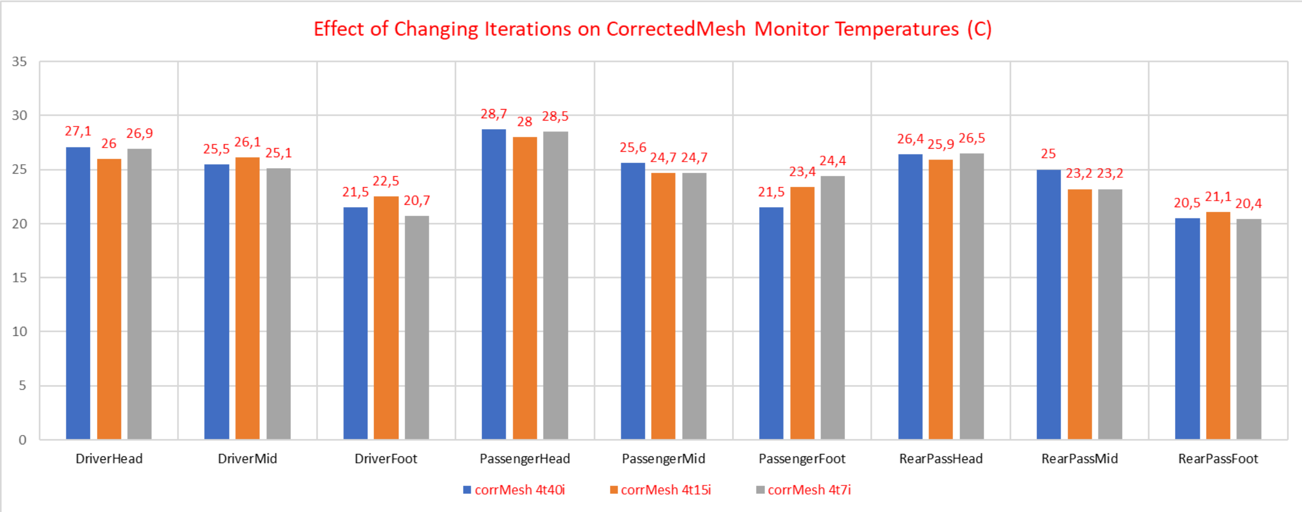


Table 4: Effect of changing iterations on temperature of large monitor volumes of improved mesh (corrMesh).

Result in the above table shows that the temperature difference in all the monitors are still less than 1C in majority of the monitors with an exception of passenger foot

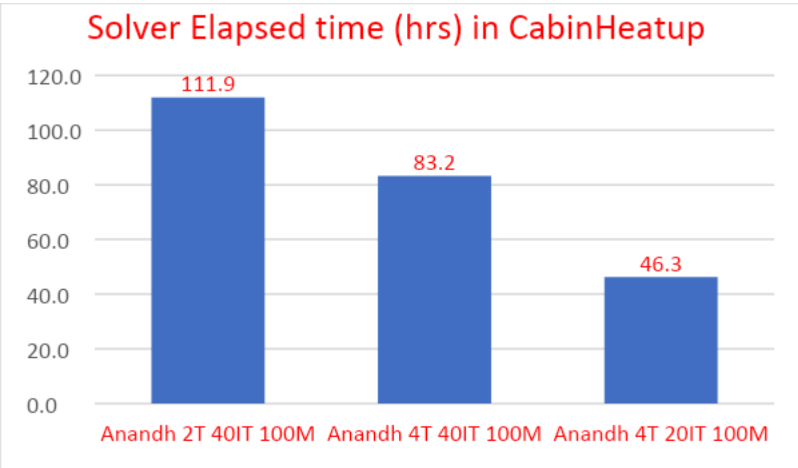


Figure 7: Reduction in solver computational time(hours) of anandhStarting Case(cabinHeatup) dure to timestep and iterations .

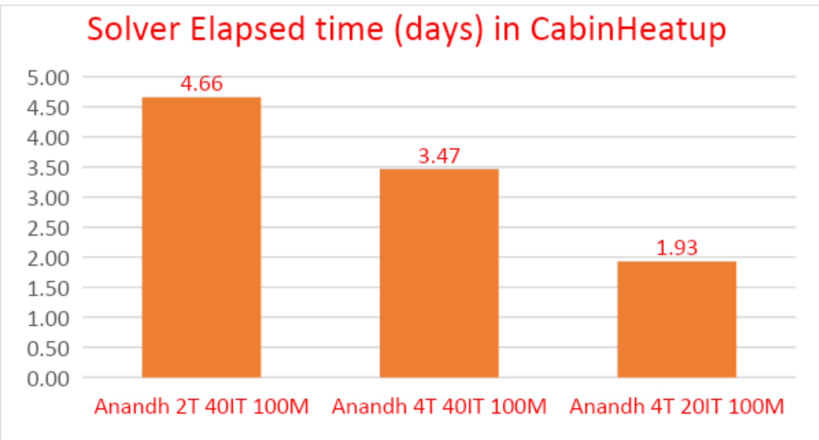


Figure 8: Reduction in solver computational time (days) of anandhStarting Case(cabinHeatup) due to timestep and iterations .

As you can see that in cabinHeatup, we have higher reduction in solver time in changing iterations study

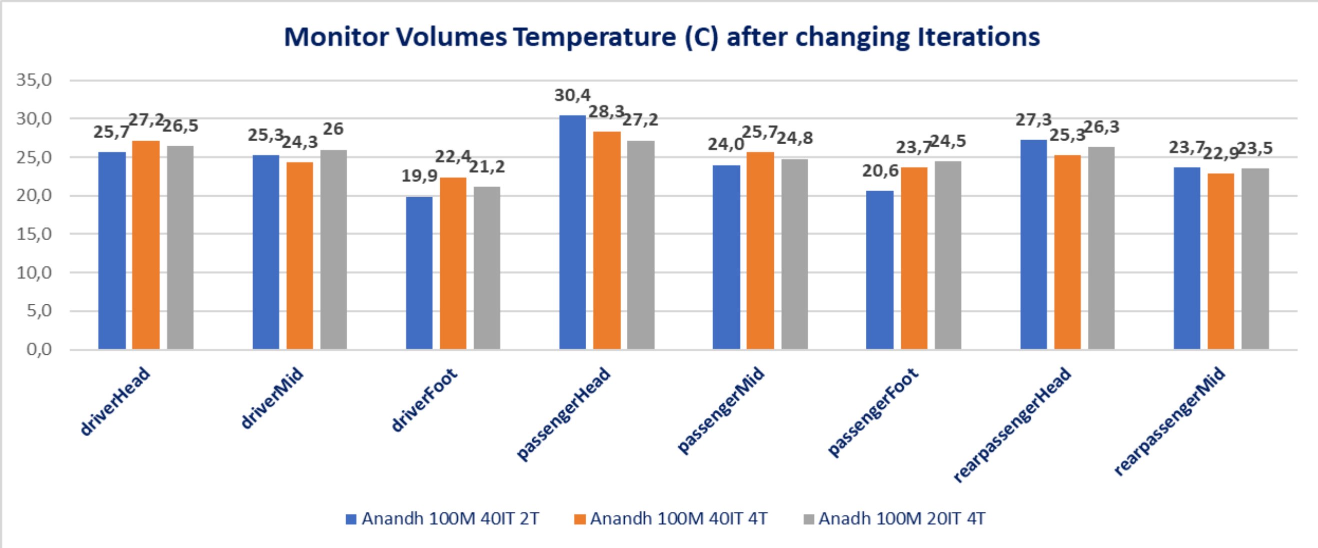


Table 5: Variation in monitor volumes temperature of anandhStarting case due to changing timestep and iterations

In the above plot, you'll see higher difference between monitor volumes from base settings, it is because of using small volumes instead of large volumes. Initially I used small volumes for comparison but the difference was more than our threshold limit (1C). The problem is that temperature plots have a lot of high and low spikes, in such case taking mean gives you more accurate results. That was one the motivation why I used mean of last N-samples in the later simulations.

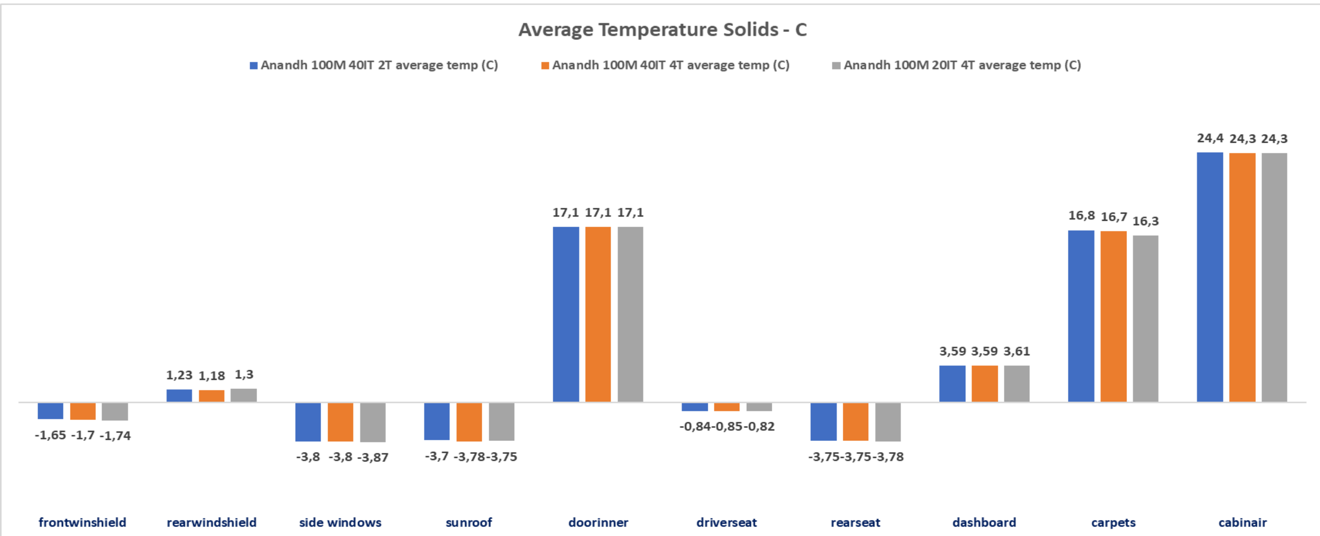
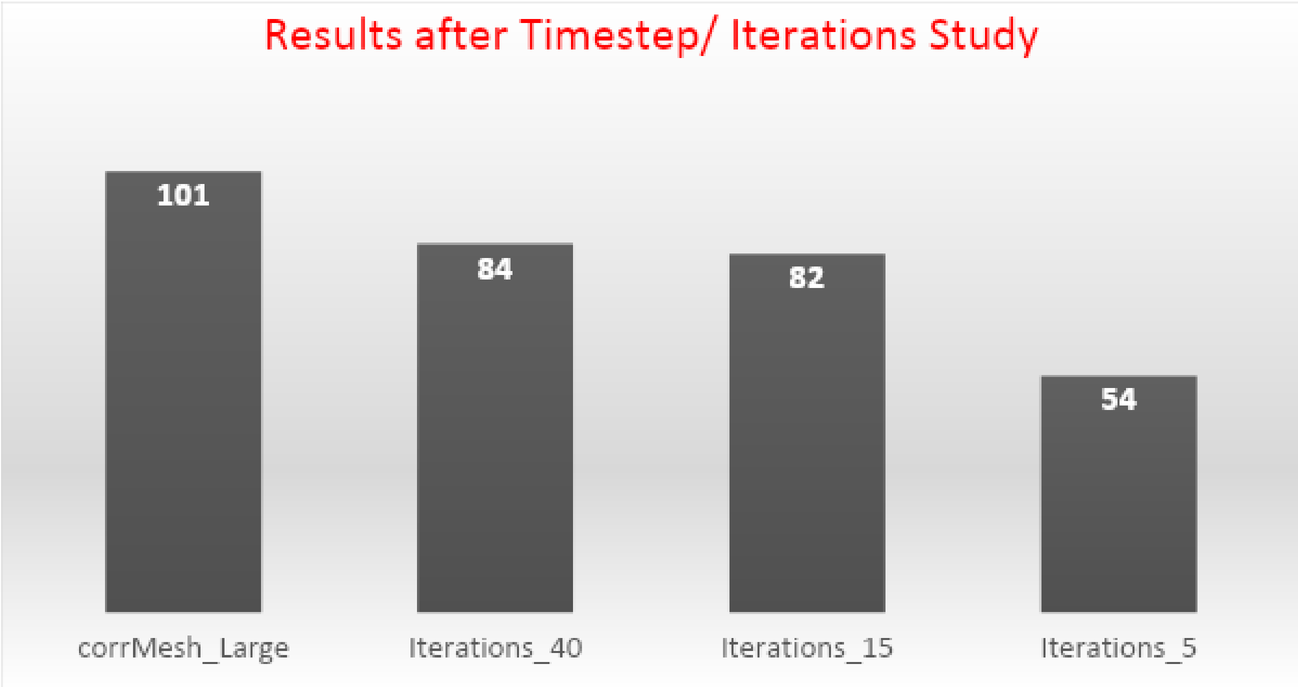


Figure 9: Variation in solids temperatures of anandhStarting case (cabinHeatup).

The reason why we focused more the temperature of monitor volumes was because in solids, we saw a very small deviation of temperature. So it was not a good measure for comparison.

So far we have reduced our solver computational time from 115 hrs to 46hrs for cabinHeatup (Anandh Starting case) while in corrMesh model we achieved a reduction from 101hrs to 56hrs. In both cases the deviation of temperature from starting case in majority of the 9 monitors is below 1C. The evaluation is based on large volumes and taking mean of last n samples.

Mesh Study



After the timestep/iterations study, the next step was to calculate the effect of mesh settings on reducing solver time. Our idea was to analyse all the parameters within mesh settings such as target surface size, minimum surface size, surface growth rate, volume growth rate, tet size etc.

In each case, one parameter Anandh settings was doubled while all the other parameter was taken same what anandh used. For example. In case 1, we increased the minimum surface size from 1mm to 2mm so all the remaining setting remains same as what Anandh used except the minimum surface size.

Total 5 case studies were performed in which first we changed the minimum surface size and target surface size of both cabin and solids (case 1 to case 5) then in case 6 and 7 mesh settings was kept same but prism layers were decreased. (from 10 to 6 in surface control and 5 to 3 for everywhere else). Below we have list of all the cases analysed

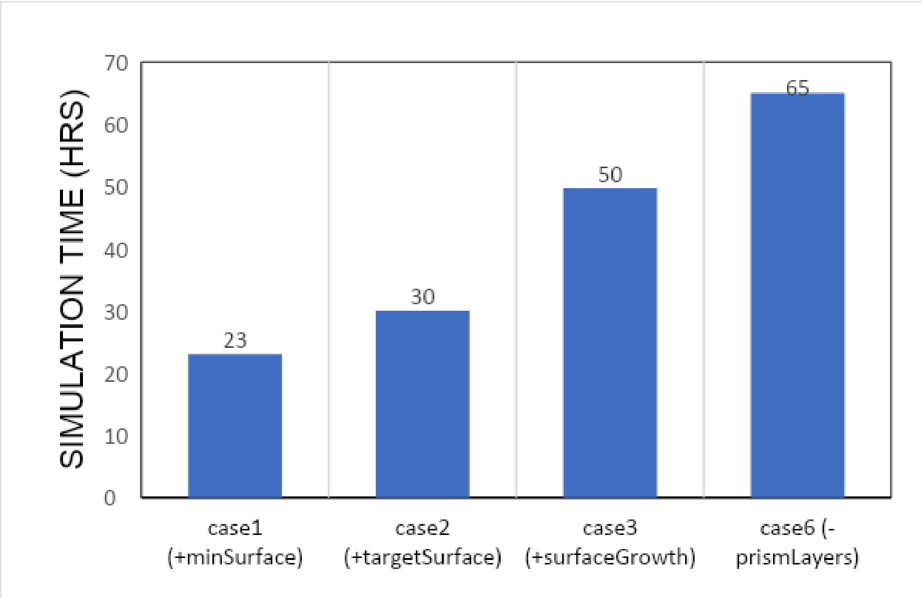
Changing different mesh parameters

Higher Surface Growth Rate
(1.2 => 1.5)

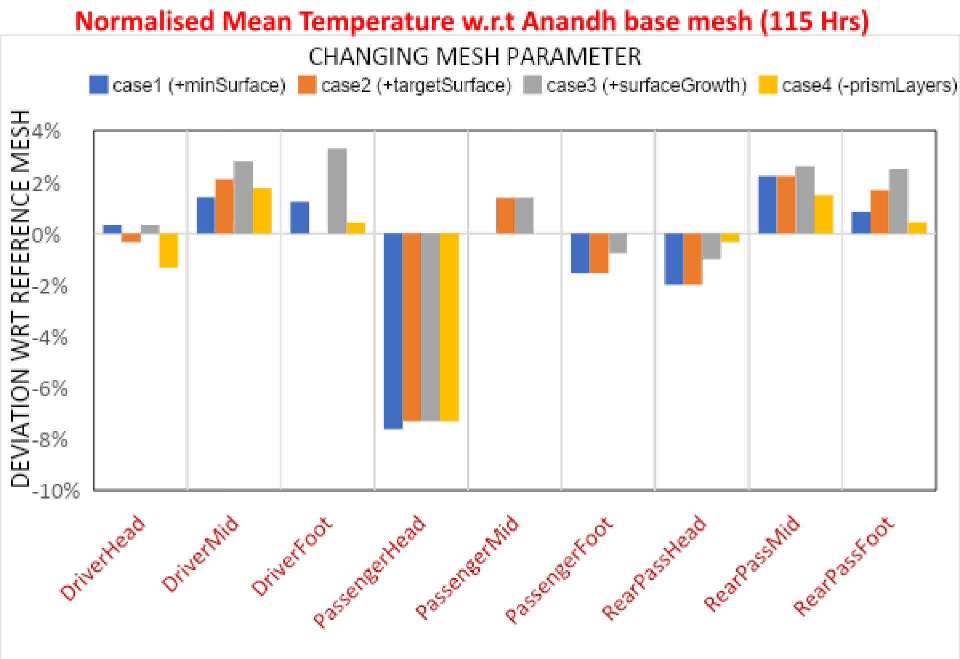
Increase Min Surface/Target
(1mm => 4mm, 4mm => 8mm)

Higher Volume Growth
(1.2 => 1.3)

Higher Tet size
(16mm => 32mm)
Lower Prism Layers
(10 => 6)



Results show that among all the cases analysed, in case1 (increasing the minimum surface size) give us the least simulation time of 23 hrs but it would be important to know how much deviation is seen in temperature values of all the 9 monitors.



Comparatively we see higher deviation than both timestep and iterations but since our objective is to reduce the computational so deviation is still under 1C. because of this we would proceed with the case in which we have the max reduction in time. (Case1). So our computational time has been reduced from 115hr to 23hr using 920 cores.

Figure 11: Setting details of different meshSettings

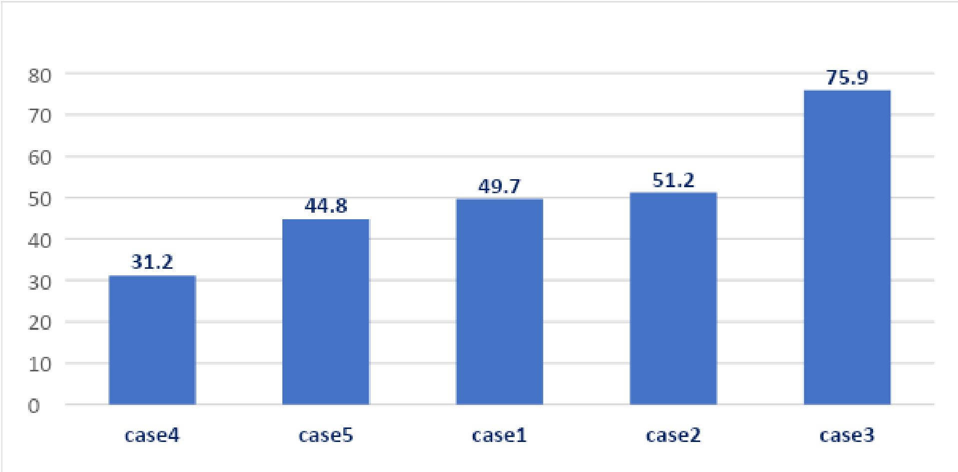
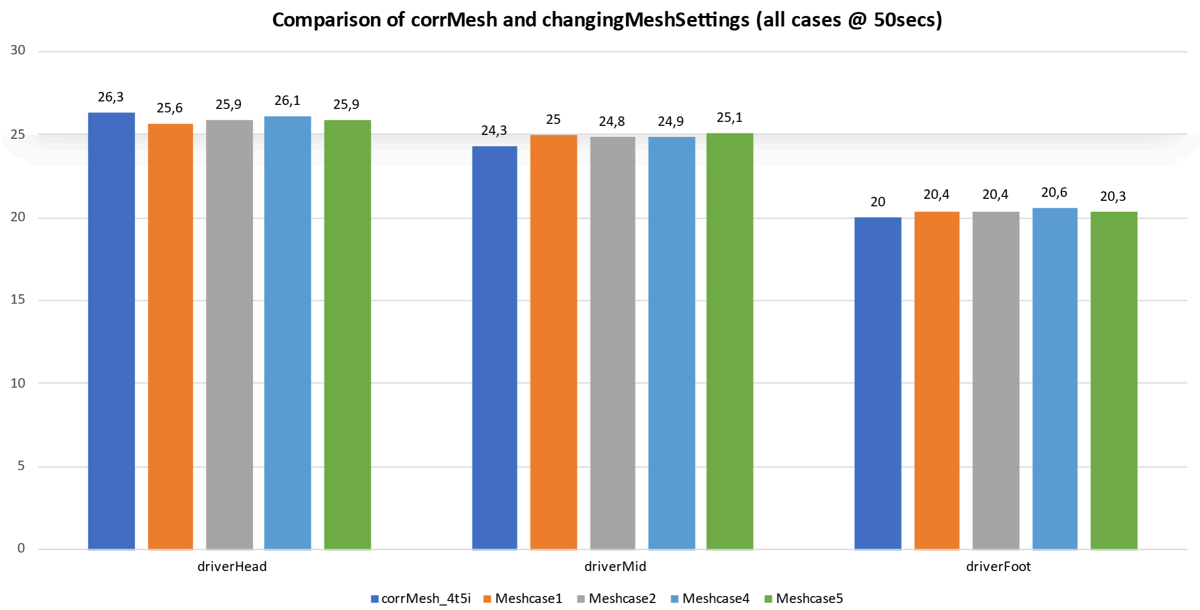
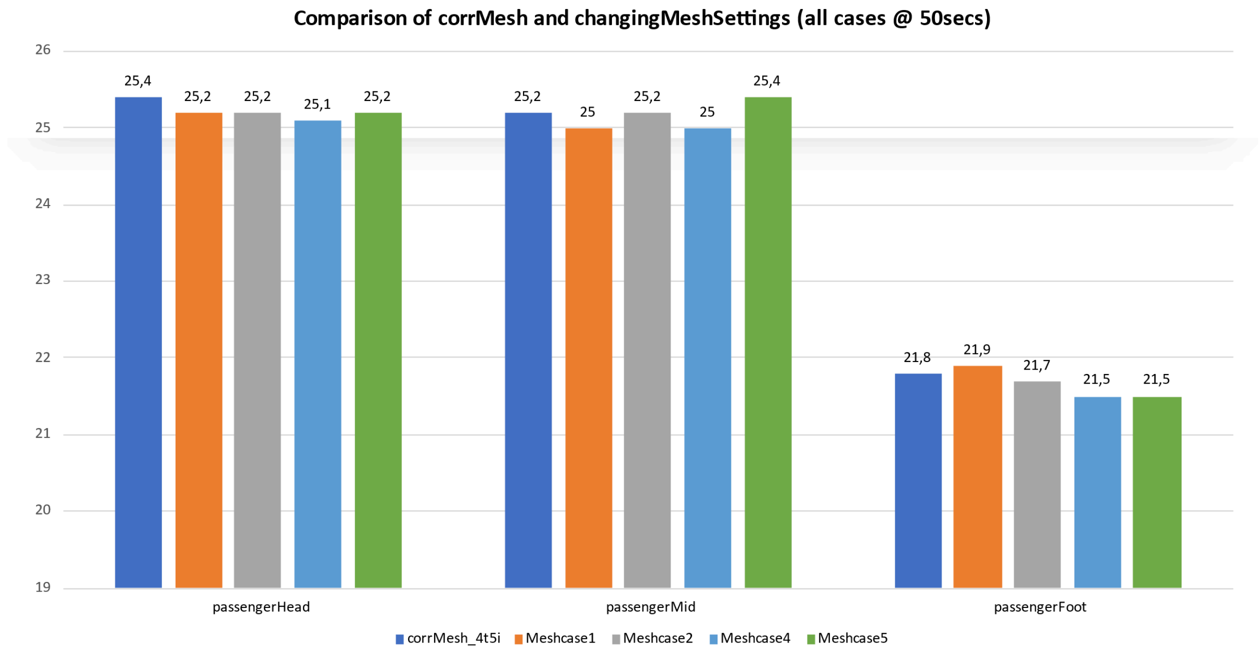


Figure 12: Reduction in no of cells in cabin due to changing the mesh settings (case 1-5)

Among all the cases that were run, case 4 was the most efficient one in which the solver time further reduced from 54 hours to 23 hours with minimum deviation of less than 0.5.





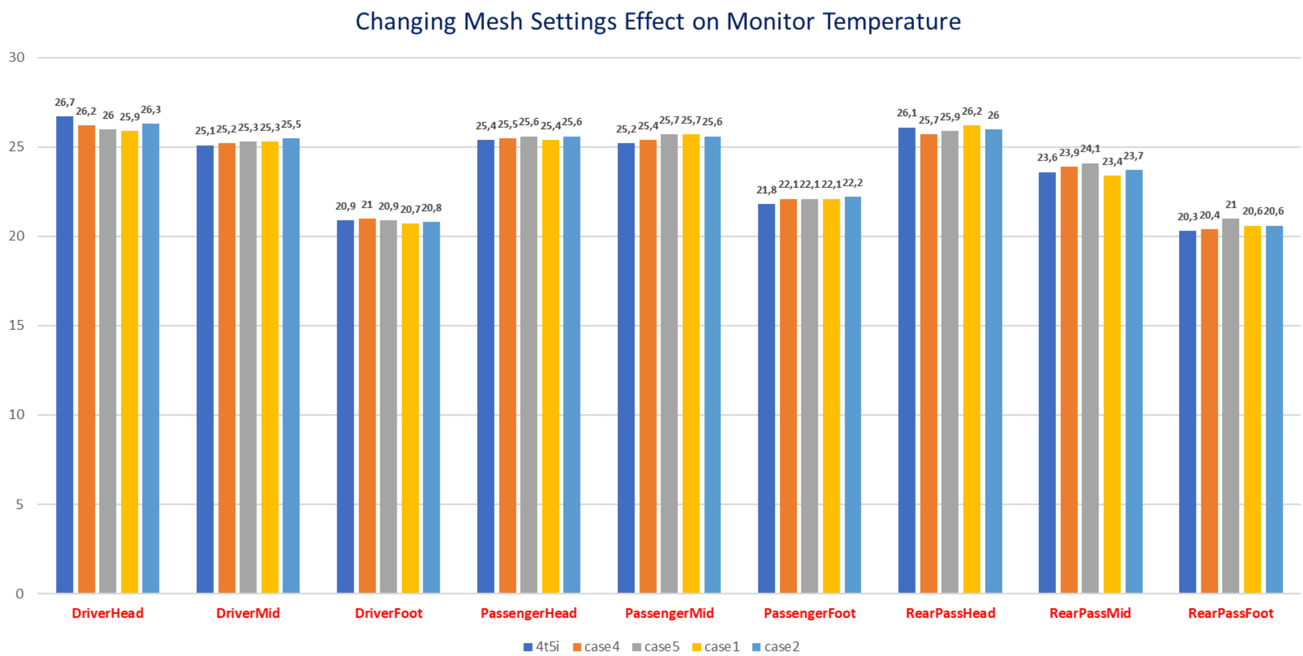


Figure 14:Monitor volumes temperature comparison of CorrMesh base case with all the cases studied in meshsettings study (case 1-5)

Case 4 turned out to be the best case among different mesh study. So it was the selected as the final case. The solver time was reduced from 115 hrs to 23 hrs with a deviation in majority of the monitors less than 1C. The reports used were at a mean of 50 secs (2500 timesteps)

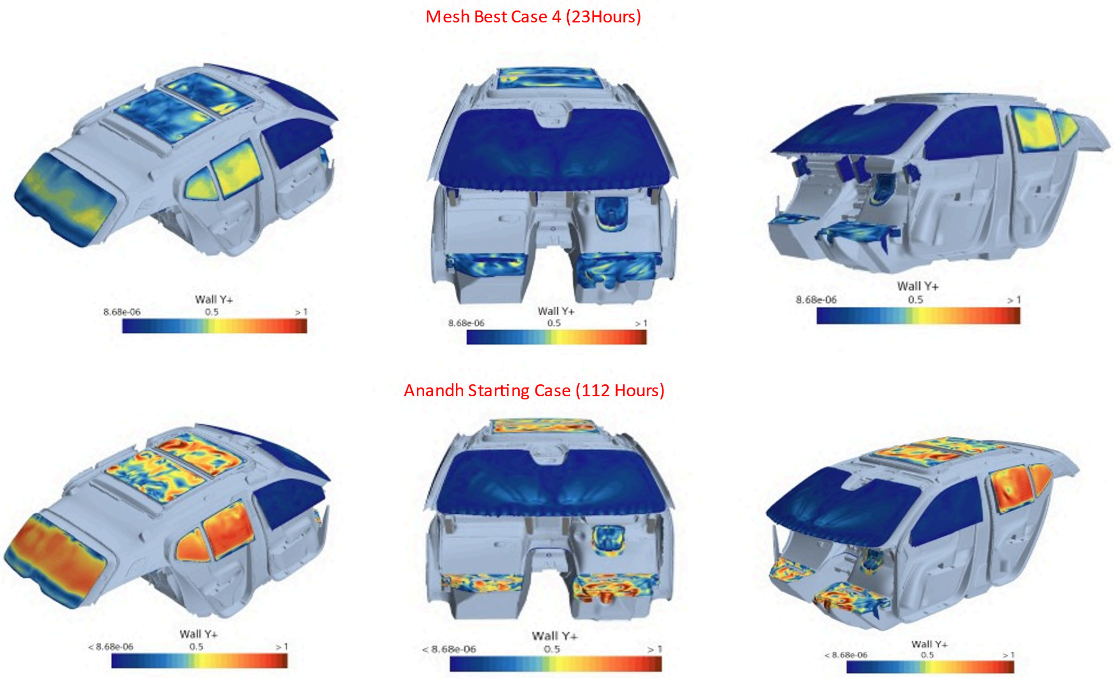


Figure 14:Wall Y+ comparison of both Anandh starting case(115hrs) and our final best case (23 hrs)

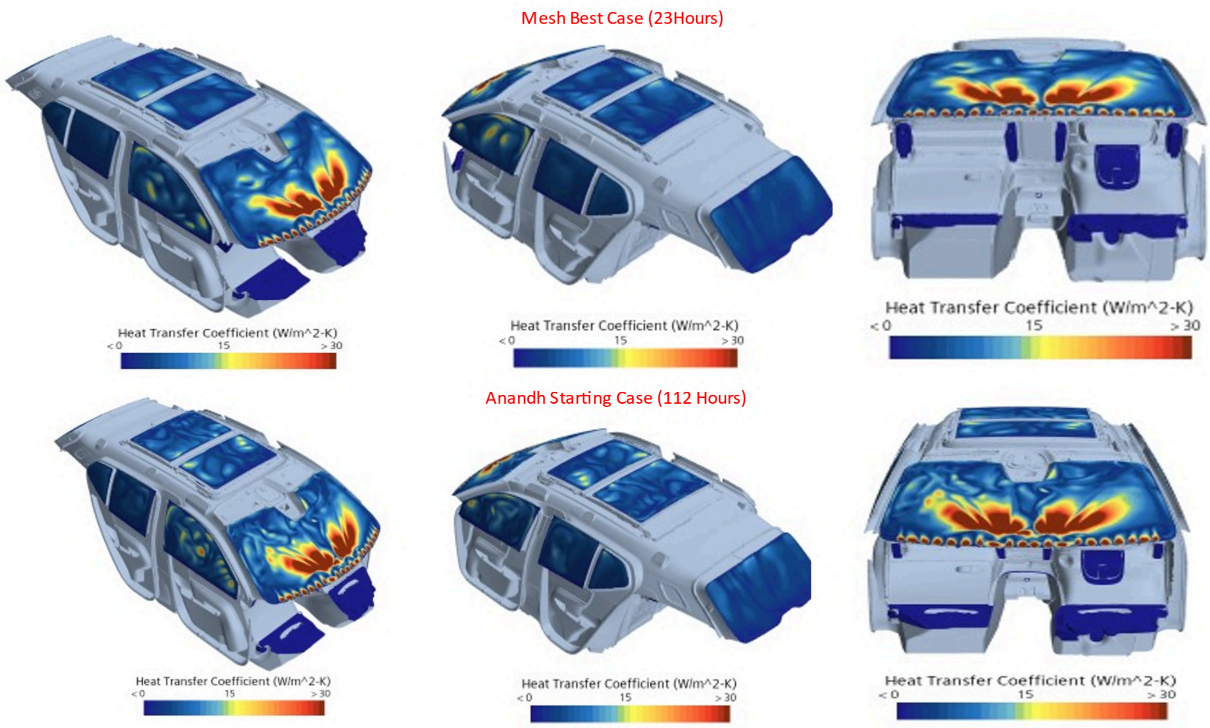
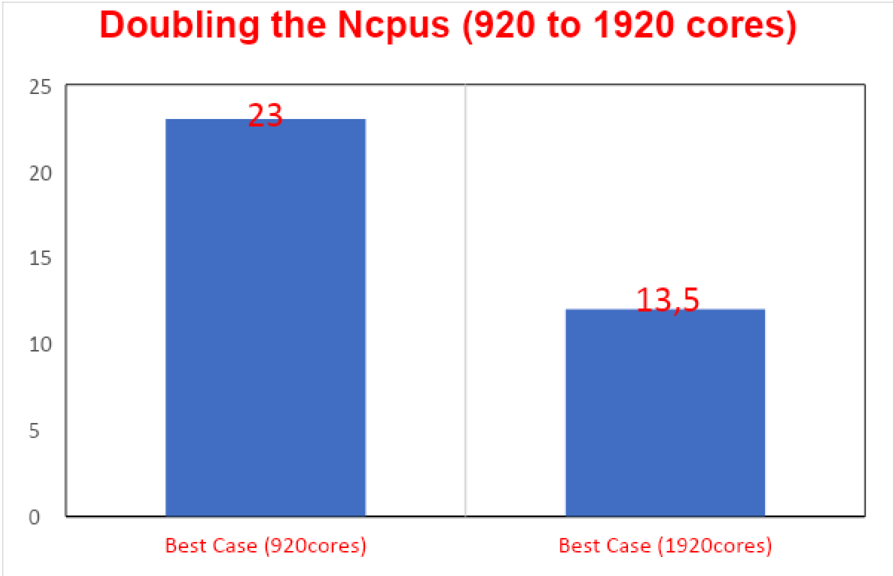
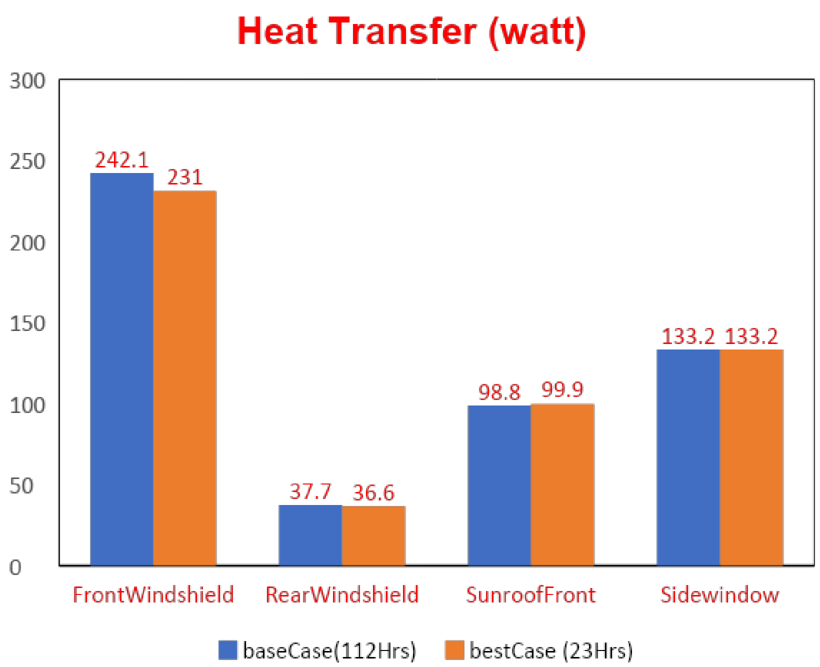


Figure 14:HTC comparison of both Anandh starting case(115hrs) and our final best case (23 hrs)

Doubling the no of cores.

Initially we didn’t increased the no of cores because Anandh simulation was getting stuck during the updating of interfaces. But in our improved model, later we increased the no of cores and saw that it was working. By doubling the no of cores from 920 to 1920, we can reduce this time to less than 13 hrs.



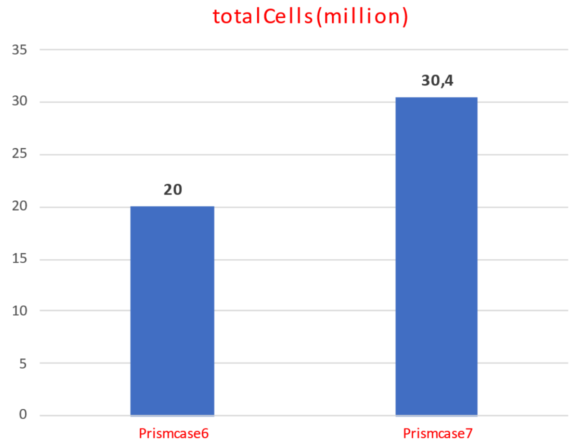


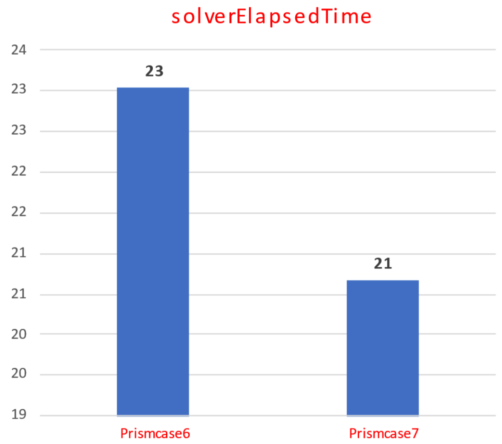
Comparison of heat transfer results between starting case (115hr) and final case shows a reduction in heat transfer in front and rear windshield which is a good sign. In sunroof our heat transfer increase while side windows shows same result.

Prism Layers Study

cabin air					
cabinMesh	minSurface	targetSurface	totalCells(million)	prism layers	controlPrism_layers
Prismcase6	4mm	8mm	20	3	6
Prismcase7	2mm	8mm	30,4	3	6

Comparison changingPrismLayers (all cases @ 50secs)





After the mesh study, another study of prism layer was done to see how reducing or increasing prism layers will contribute to the final results. Two case of prism layers was used and in each one we saw a smaller decrease in computational time.

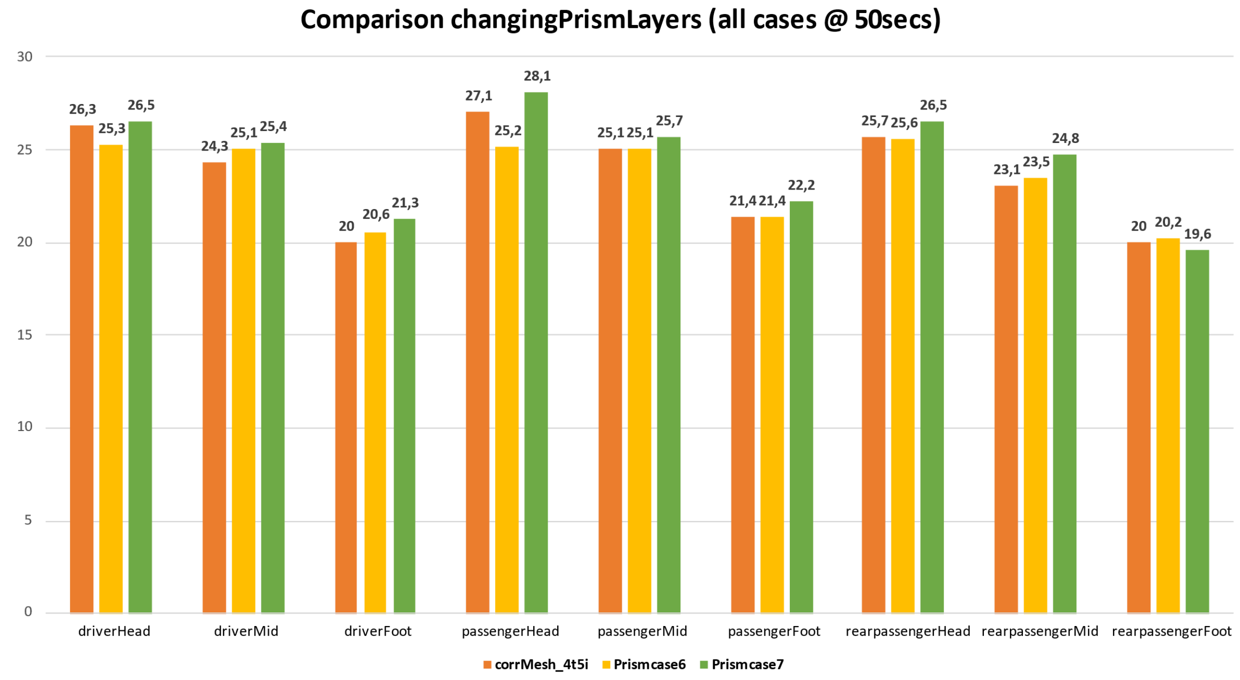
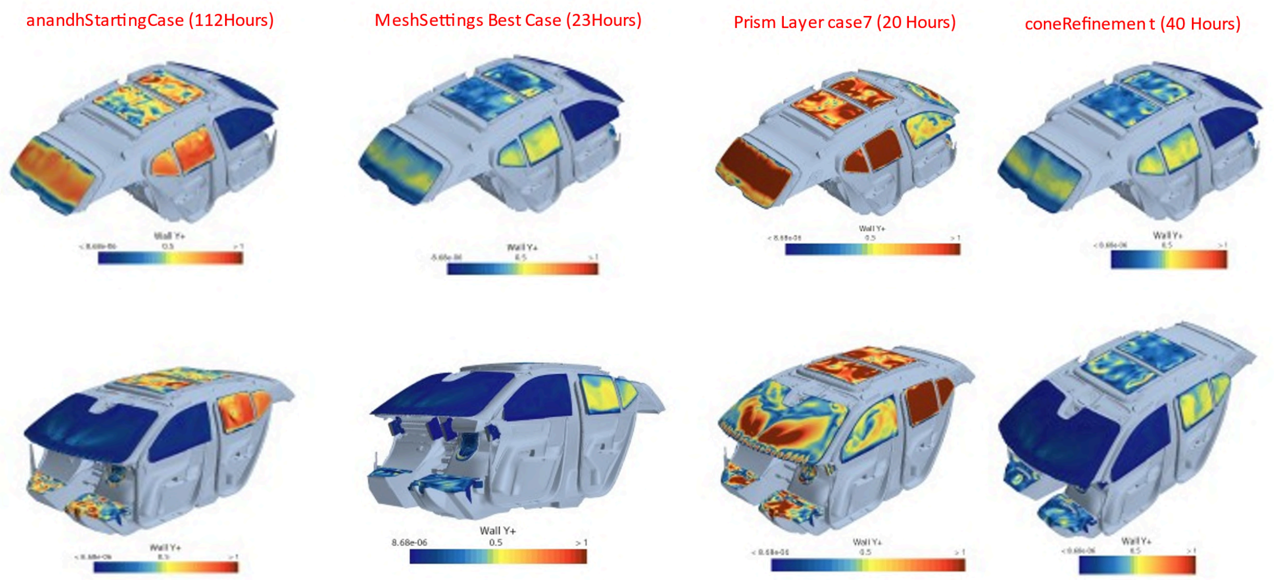


Figure 16:Comparison of CorrMesh with changing prism layers cases



The study of prism layers was discarded because of significant deviation in temperatures of monitor volumes. Also if you look at the Y+ comparison, we are seeing a major portion of area above 1 which is not so good compared to other cases. All this significant deviation came at an expenses of smaller reduction in computational time. Considering that, this study was discarded.

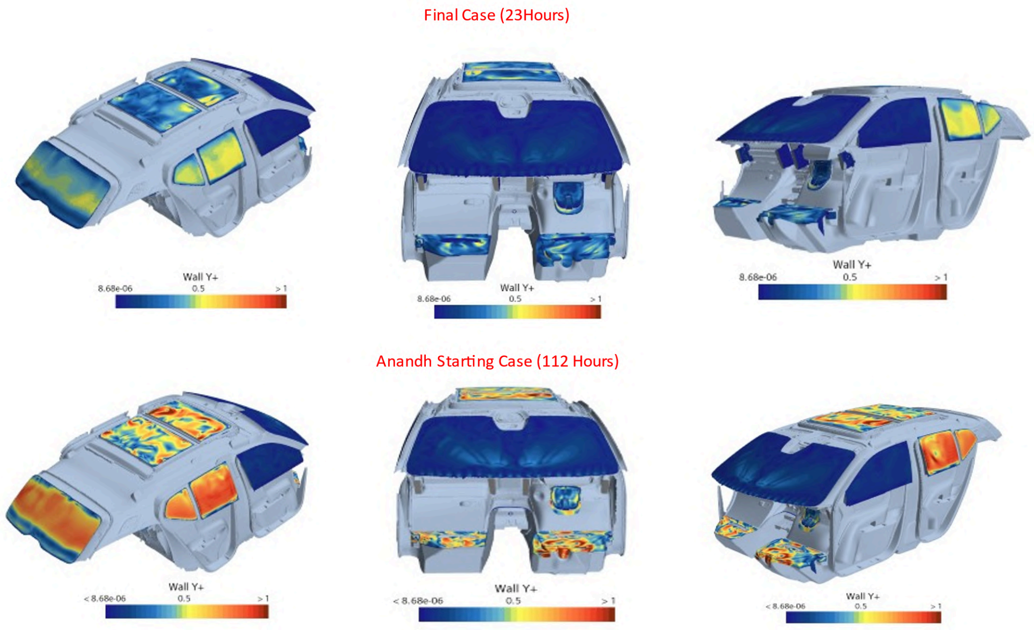


Figure 18:Wall Y+ scenes comparison between anandhStartingcase(112 Hrs) and corrMesh final case (23hrs)

First comparison of Wall Y+ between the Anandh Starting case and the final case. CorrMesh shows more area having y+ values less than 1 especially around areas where we need our boundary layers to be captured accurately.

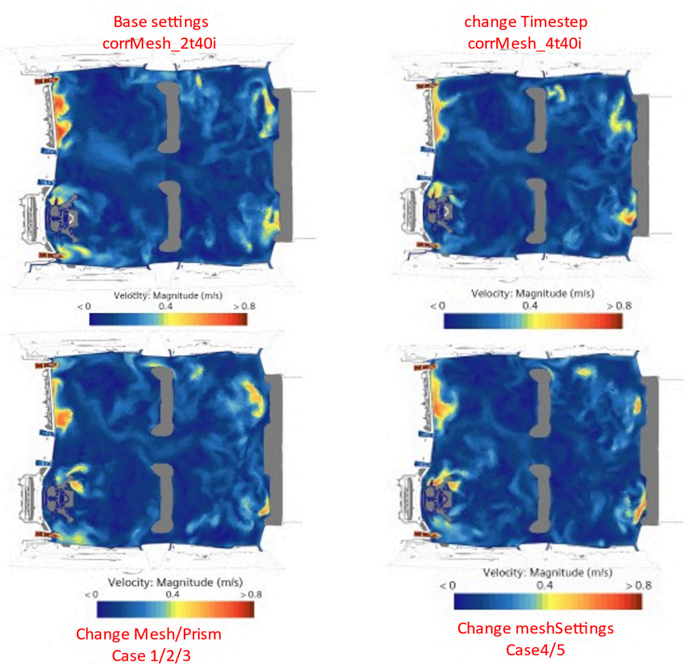


Figure 24:Comparing cabin air velocity profile (Z 110) of all the the studies done (timestep, iterations, mesh, prism layer, trunk)

Above figure shows the profile of cabin from top view (-X). Around dashboard we have some variations.

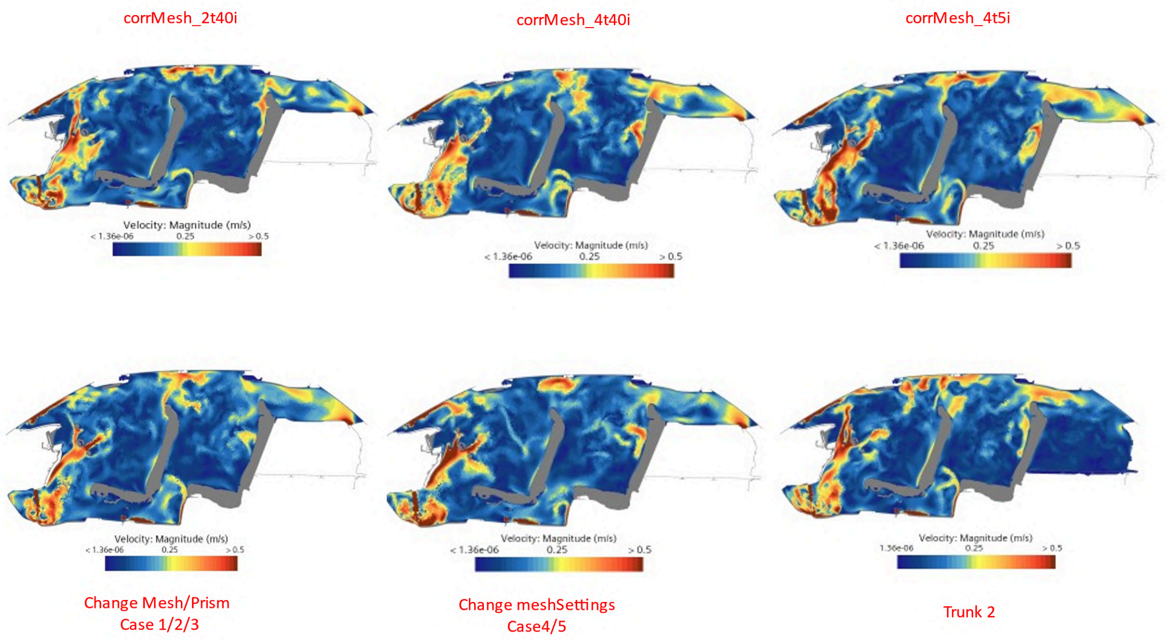


Figure 25:Comparing cabin air velocity profile (-Y -250) of all the the studies done (timestep, iterations, mesh, prism layer, trunk)

We can clearly see that coarsening mesh results in dissipation of jet. Because of that the flow cannot reach the head positions effectively as it subsides during the flow.

Climate Refinement Study

We had couple of meetings with the climate team how they do refinement for their models. They used boxes around the vents so we tried that also.

One of the ideas was that how we can have a very fine mesh around the vents whereas everywhere else we could have a coarse mesh. For that two type of refinement was done, conehead refinement and box refinement. In cone refinement, cones were created around the vents that goes towards the head position monitor volumes. While in box refinement the whole front was refined whereas everywhere else the mesh was kept coarse.

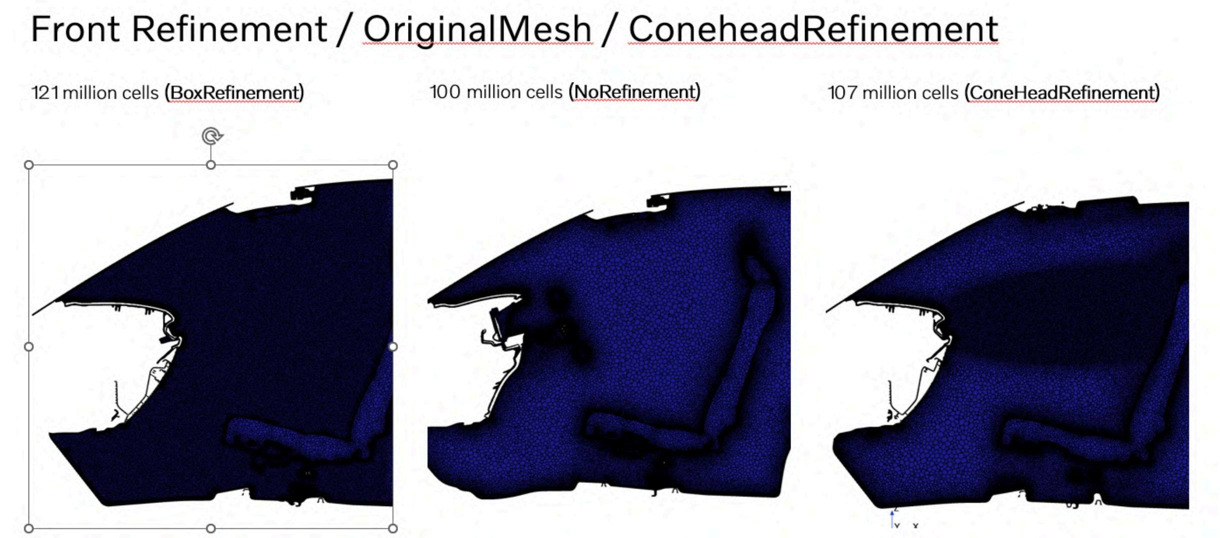


Figure 25:Side by side comparison of different refinement techniques(Box refinement/ Cone refinement) of cabin air

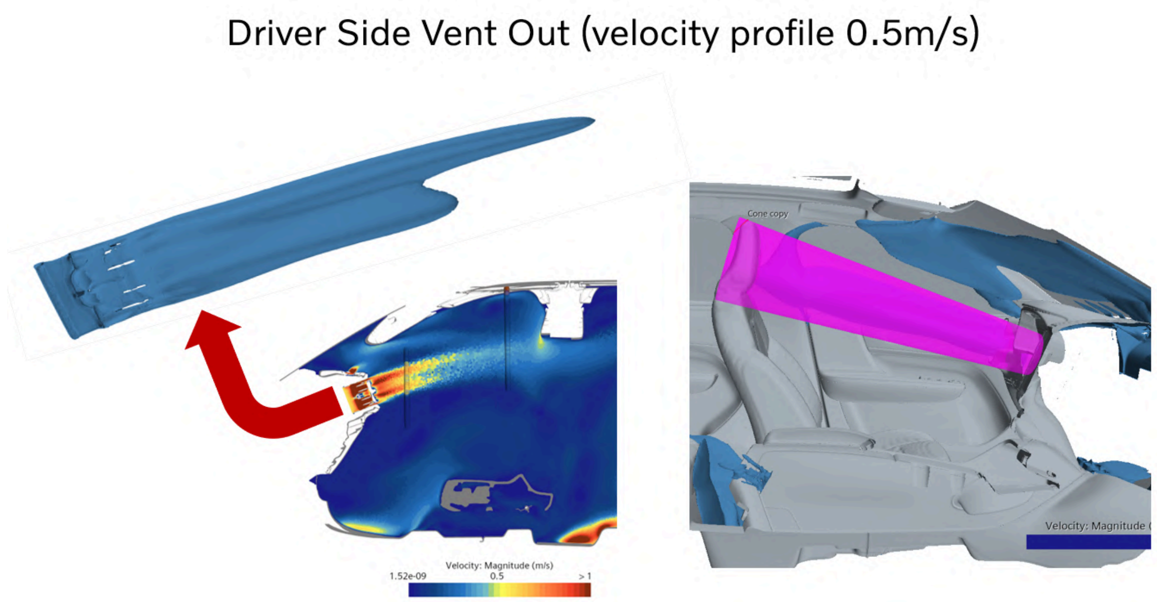


Figure 26:Velocity jet profile around the cabin vent outlet and its refinement using cones

If you look at the above figure, a cone was created from vent towards the driver position. Everywhere else the mesh was kept coarse.

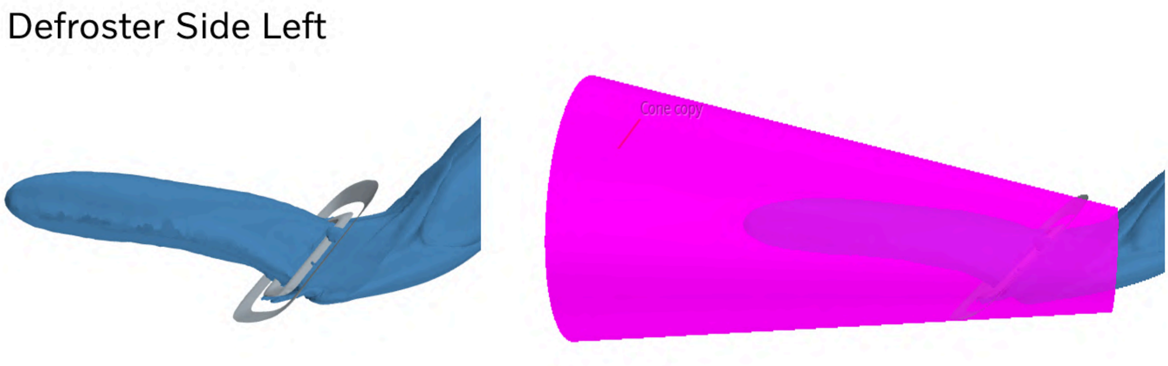


Figure 27:Velocity jet profile around the cabin defroster outlet and its refinement using cones

At each vent, similar cones were created. Similarly two line probes are positioned 10cm to vent and head position. Based on that, the flow can be compared for all the three case whether such refinement had any impact on stronger flow or not.

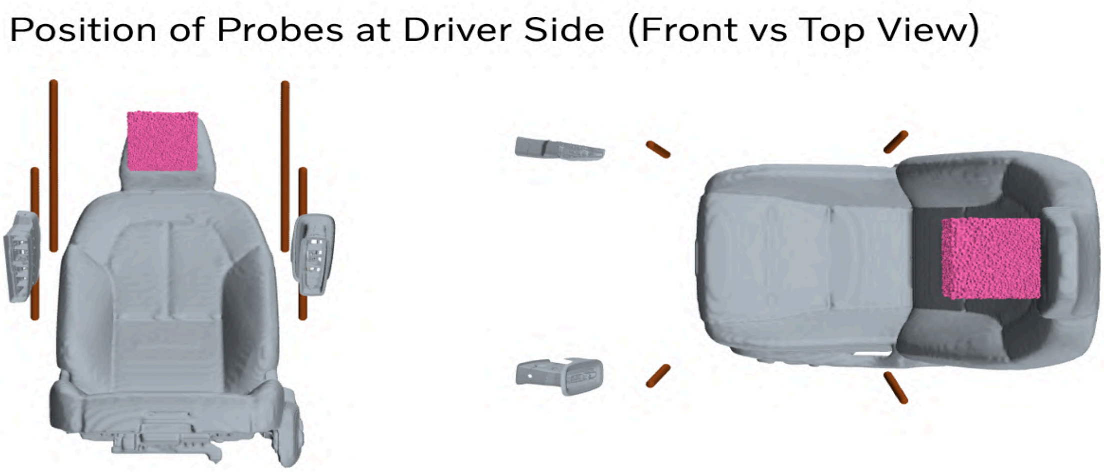


Figure 28:Positioninig of line probes to analyze jet profiles of cabin vent outs

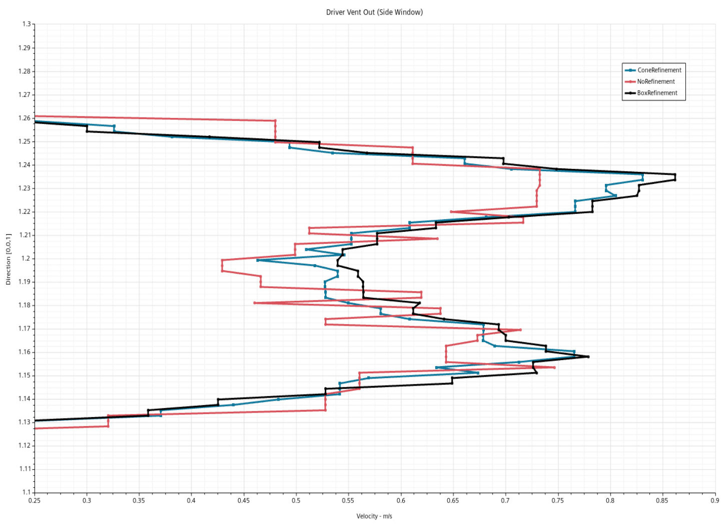
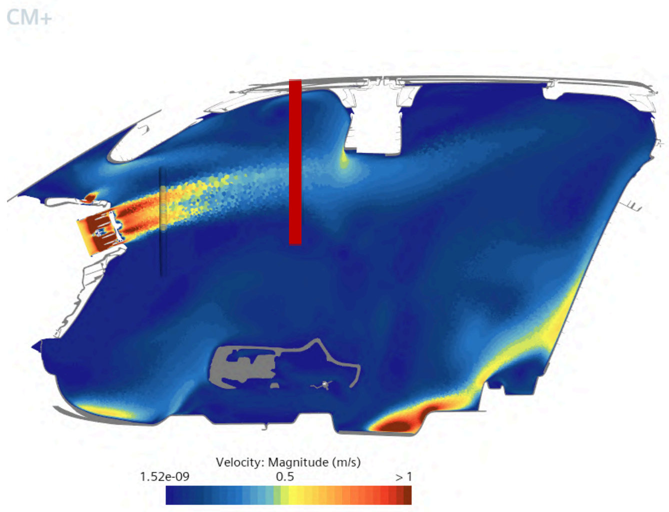


Figure 29:Velocity profile comparison of box, cone and no refinement at 10cm distance from cabinventOut

Though box refinement has better velocity profile compared to no refinement but coneRefinement is the most efficient because in box refinement, a lot of unnecessary areas also get refined.



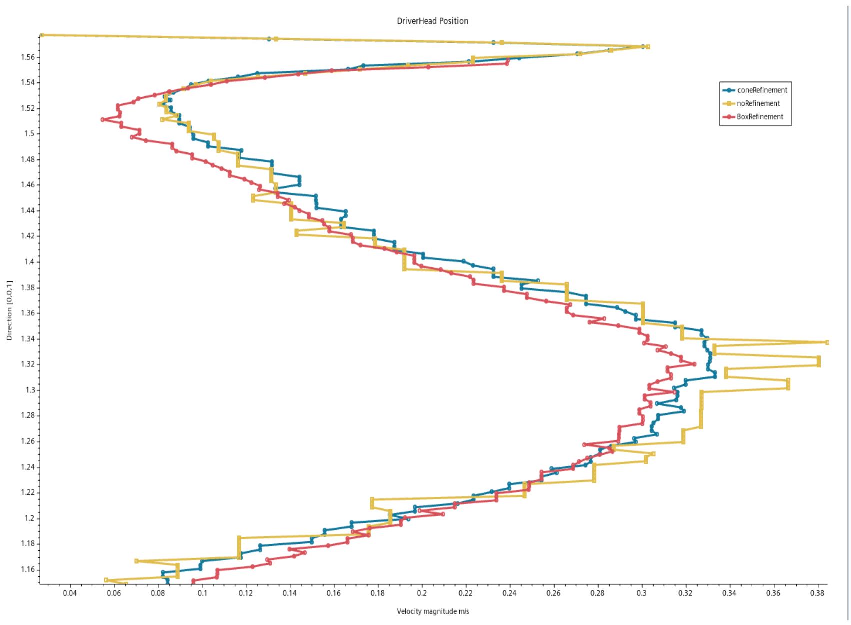


Figure 30:Velocity profile comparison of box, cone and no refinement at 10cm distance from driverHead position

At center vents, cone refinement is the best

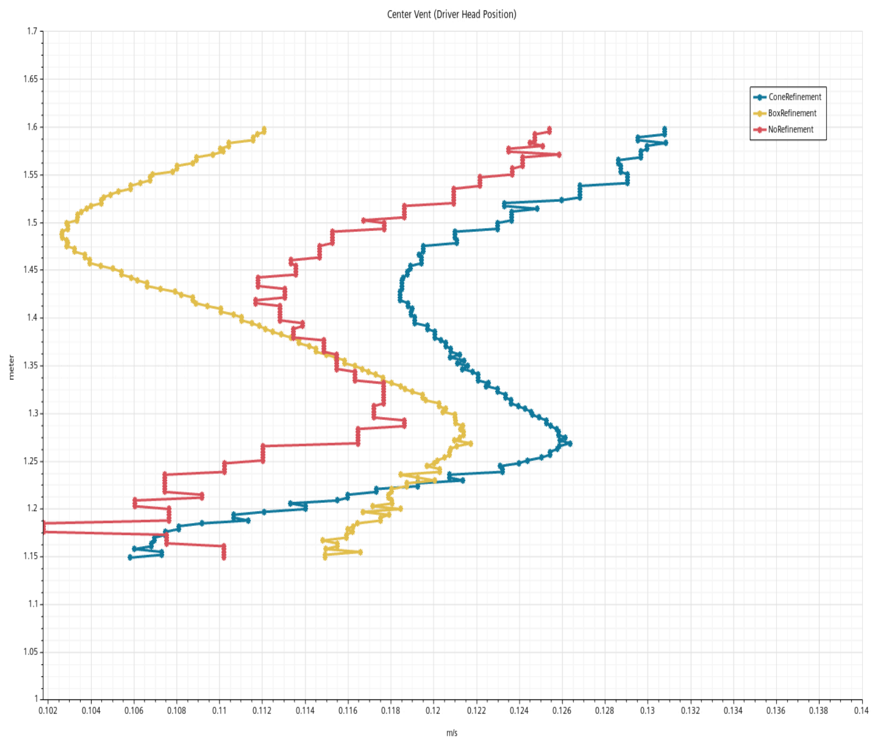
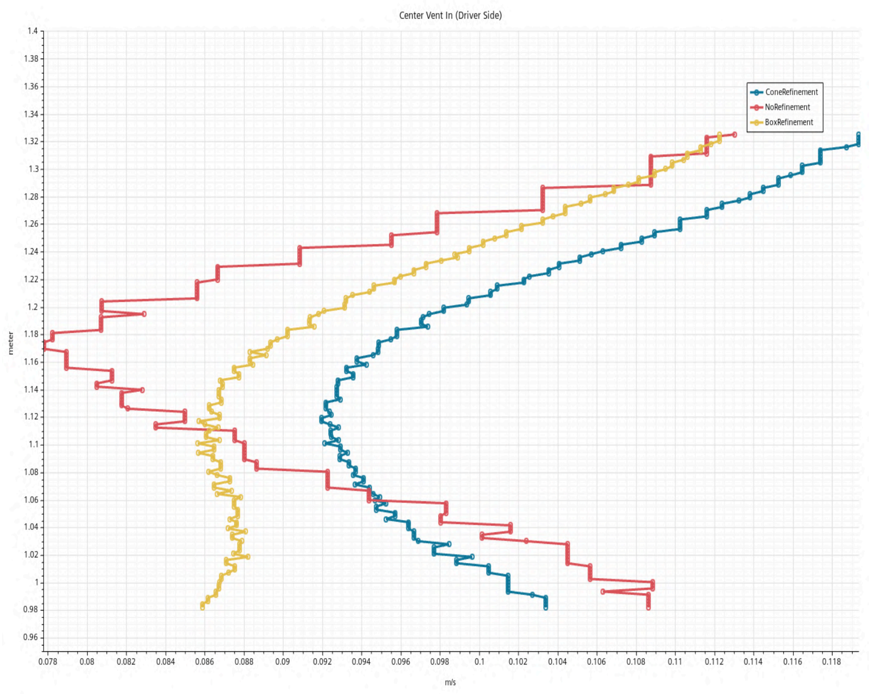


Figure 30:Velocity profile comparison of box, cone and no refinement at 10cm distance from driverHead Position



In order to see the effect of refinement technique used by climateTeam of Volvo Cars, a similar study was conducted to see how much improvement we see in the temperatures of monitor volumes.

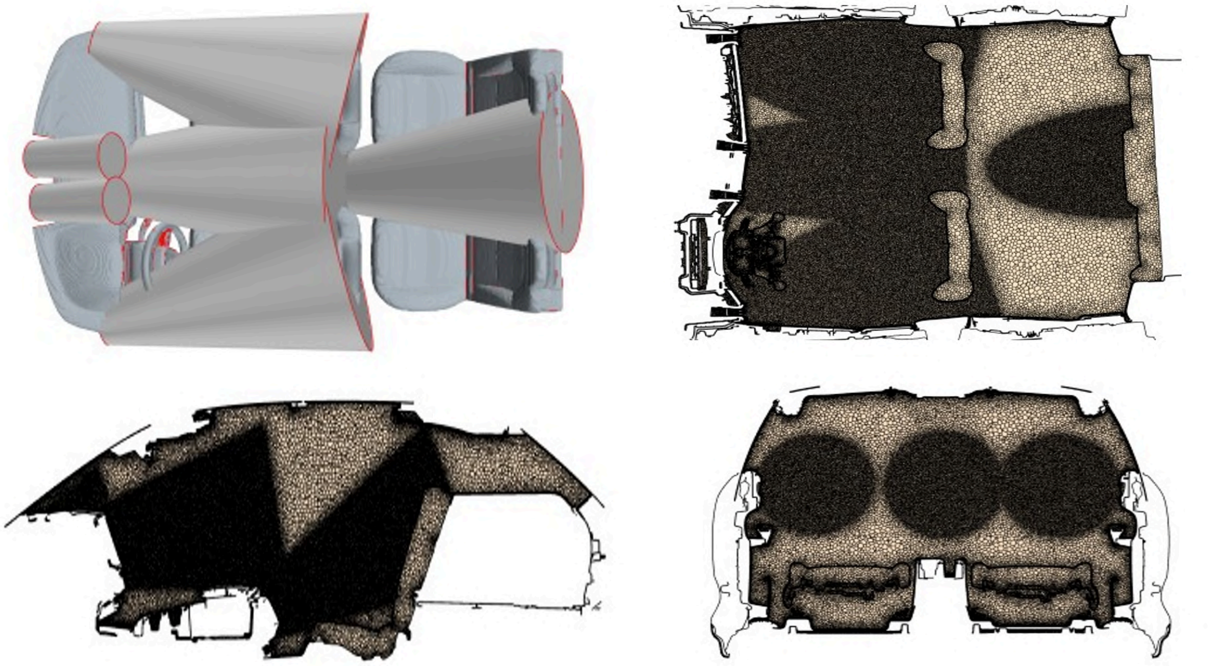


Figure 29:Cone refinement of corrMesh (40 million cells in the cabin)

Cones were created around all the vents resulting in a mesh that had 42million cells in cabin. Results are as follows

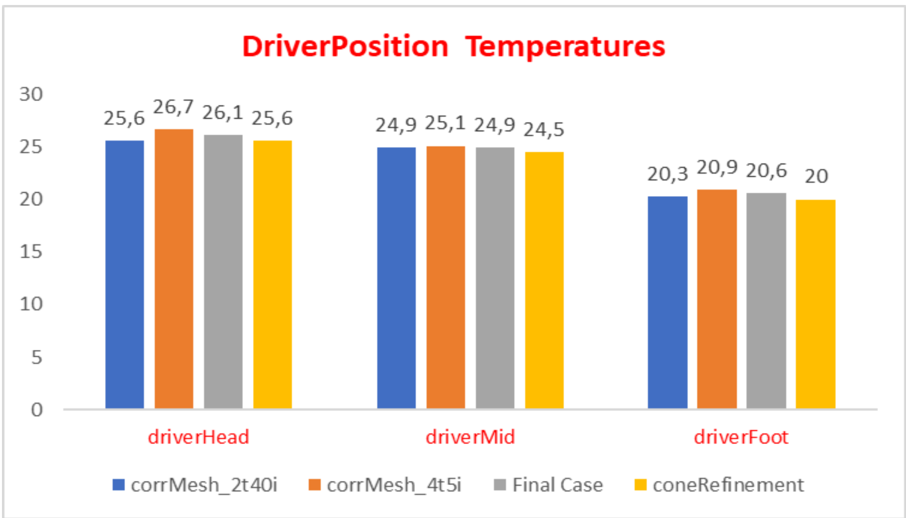


Figure 32:Comparison of driver position monitor volumes base case temperature with all the main cases (bestcase, refinement, iterations)

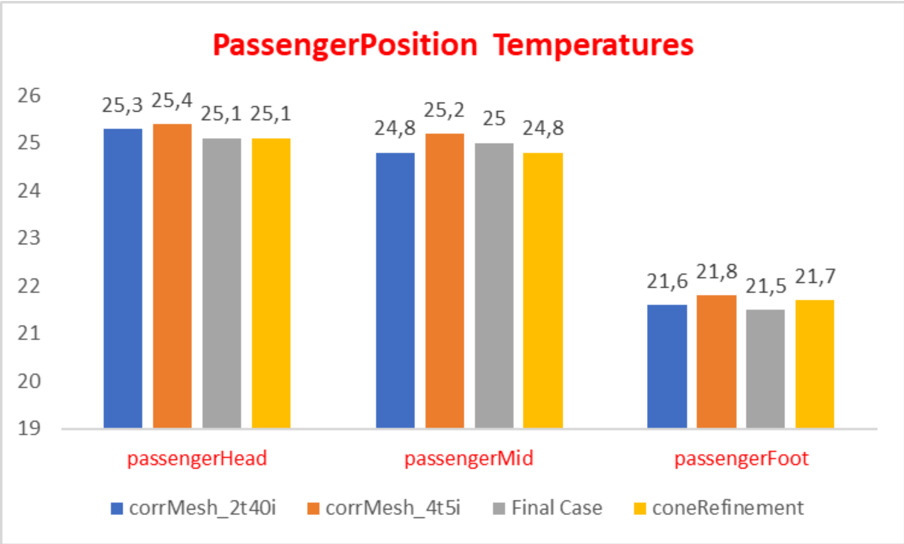


Figure 33:Comparison of passenger position monitor volumes base case temperature with all the main cases (bestcase, refinement, iterations)

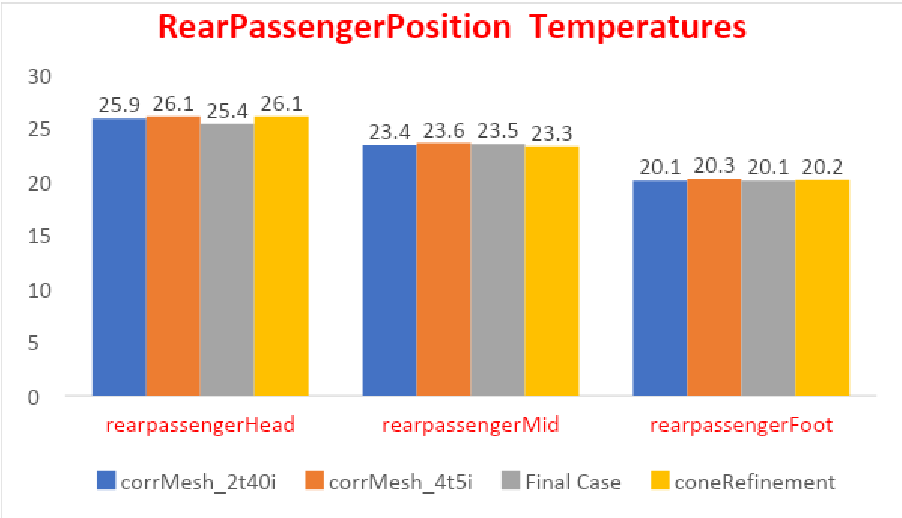


Figure 34:Comparison of rear passenger position monitor volumes base case temperature with all the main cases (bestcase, refinement, iterations)

We expected that refinement will result in increase of monitor volumes temperature and the mesh is more fine but results show that we don’t see a bigger difference except for few monitors. The no of cells has increased from 30 million for the best case to 40 million for refinement but the temperature at monitor volumes still increased or decreased by a small value. Considering that we stick to the our final evaluation.

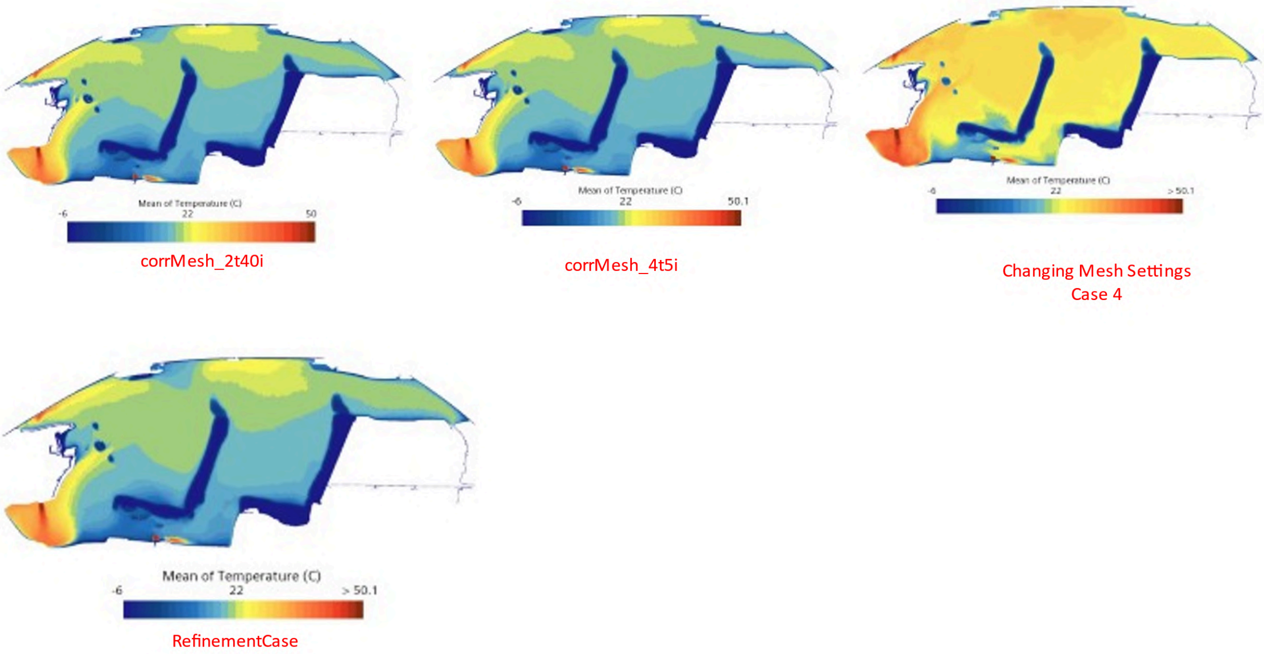


Figure 35:Comparison of temperature profile at 50 secs mean monitor(2500timesteps)

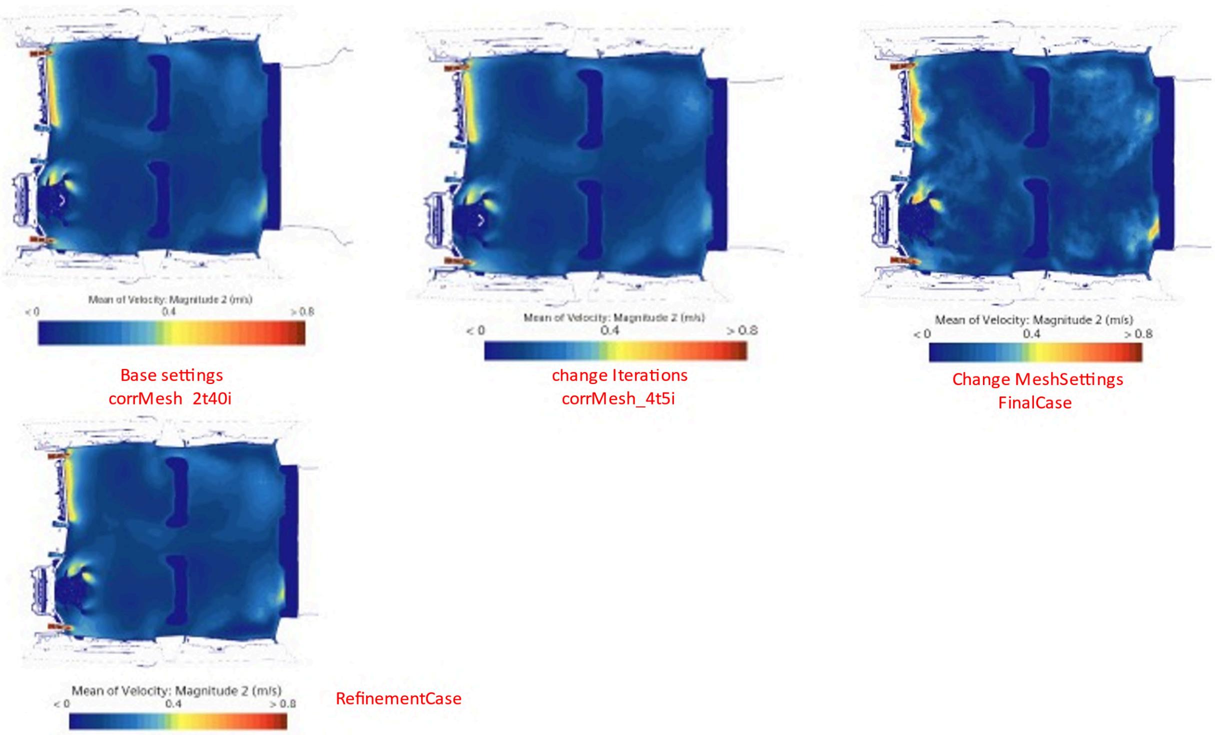


Figure 36:Comparison of velocity profile (Z-110) at 50 secs mean monitor (2500timesteps)

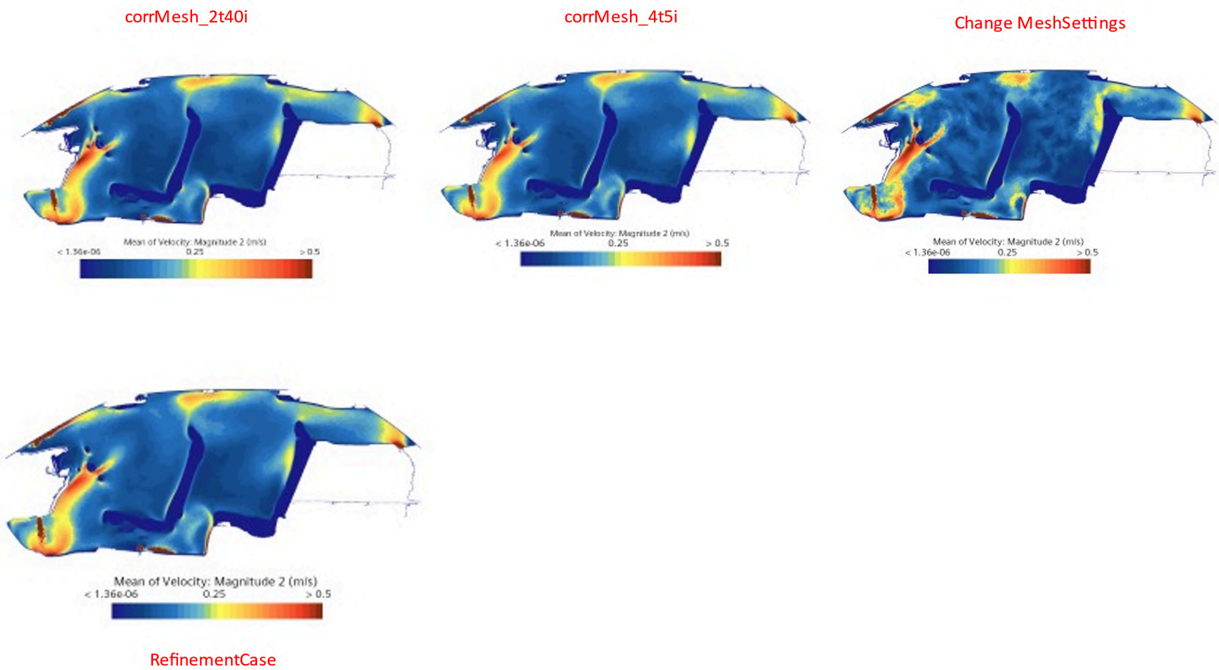
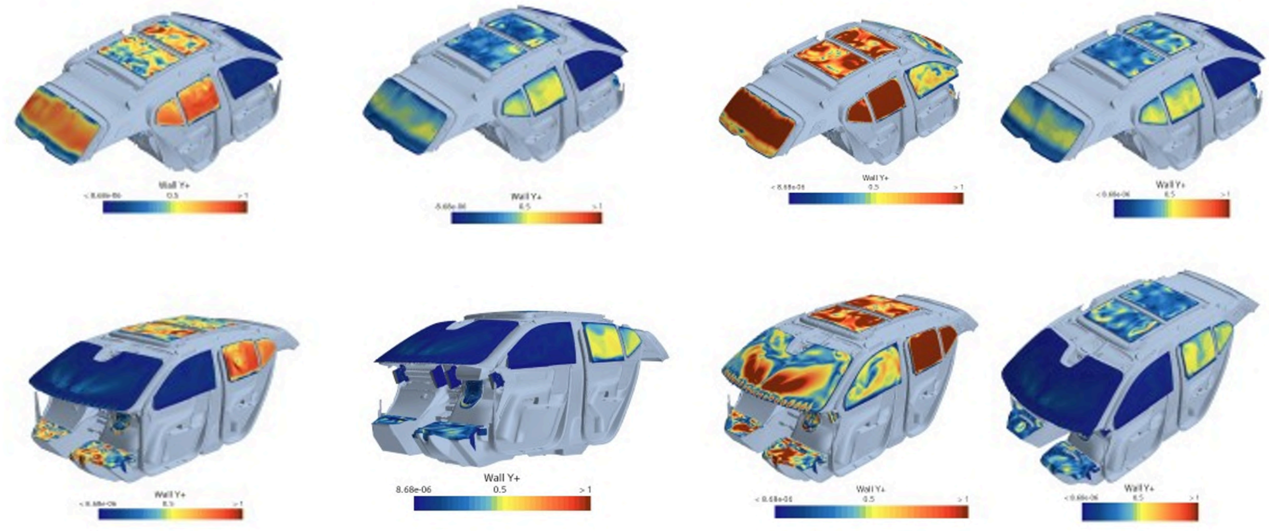


Figure 37:Comparison of mean velocity profile (Y-250) at 50 secs mean monitor (2500timesteps)

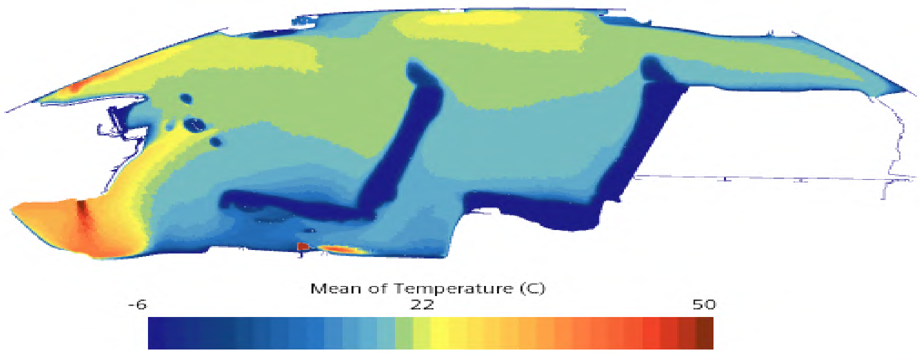
Final Result

anandhStartingCase (112Hours) | MeshSettings Best Case (23Hours) | Prism Layer case7 (20 Hours) | coneRefinemen t (40 Hours)

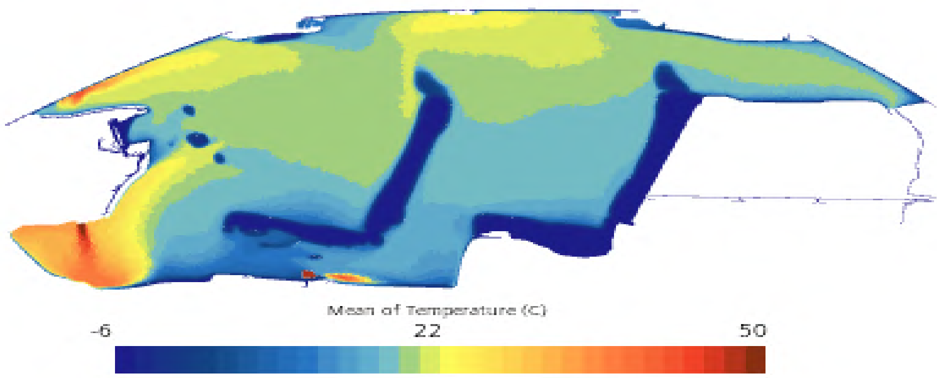


50sec Mean Temp (C)

Anandh
StartingCase
(115Hours)

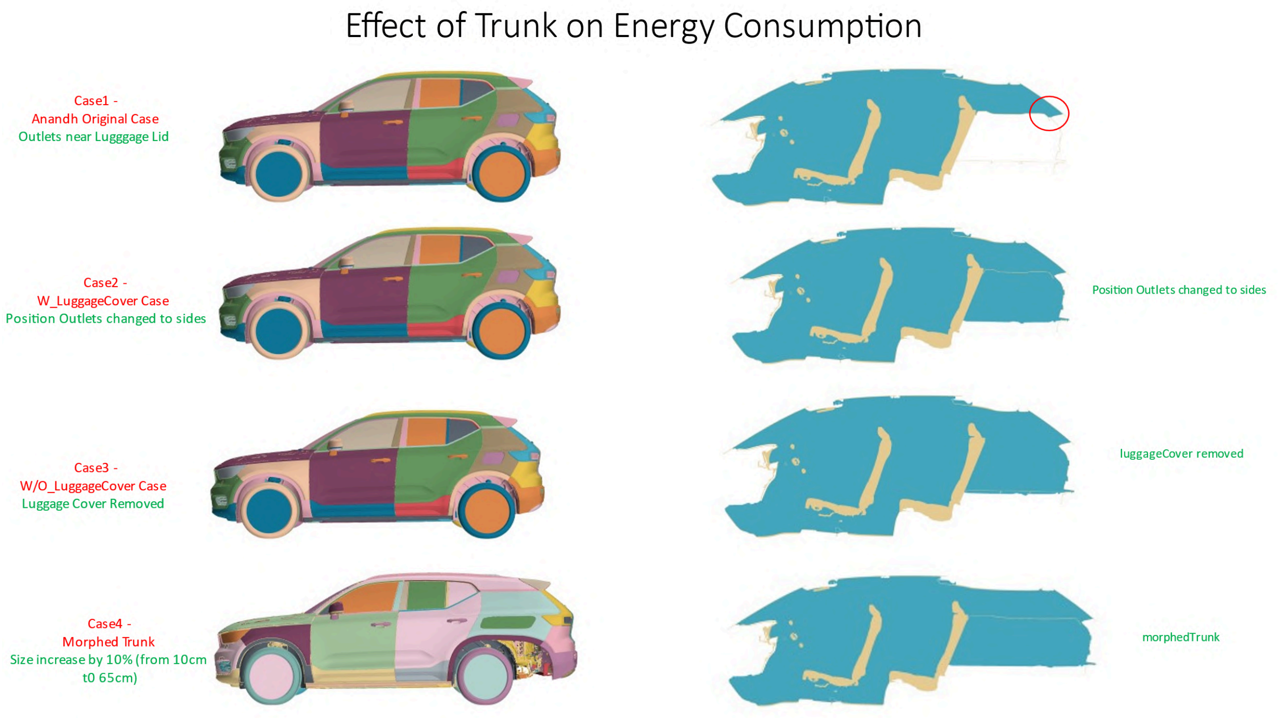


Best Case
(23Hours)



Trunk Study on Energy Consumption

Another part of my traineeship was to study the effect of trunk on the energy consumption.



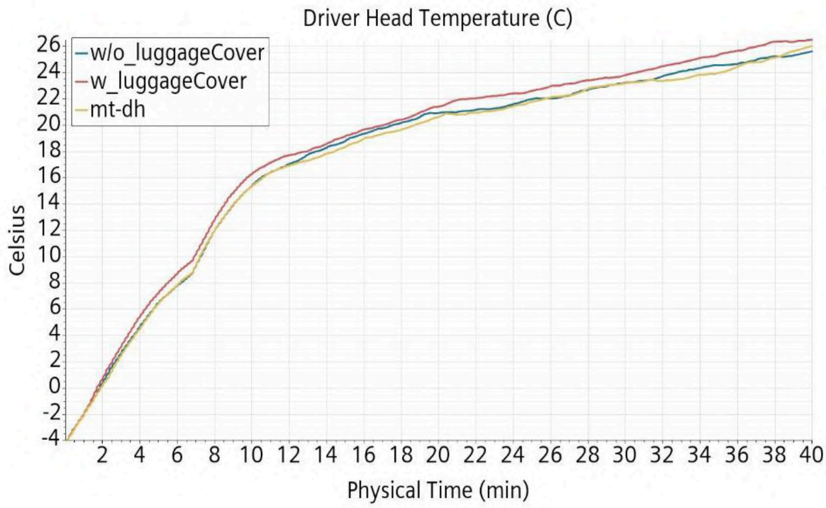
We have four cases:

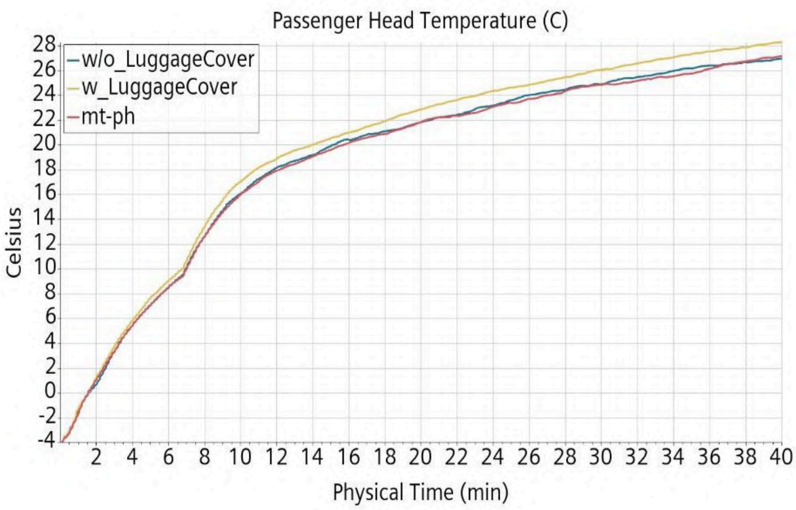
Case1 – Original case which Anandh used, in which the outlet is next to luggage cover. In this we don't mesh the trunk

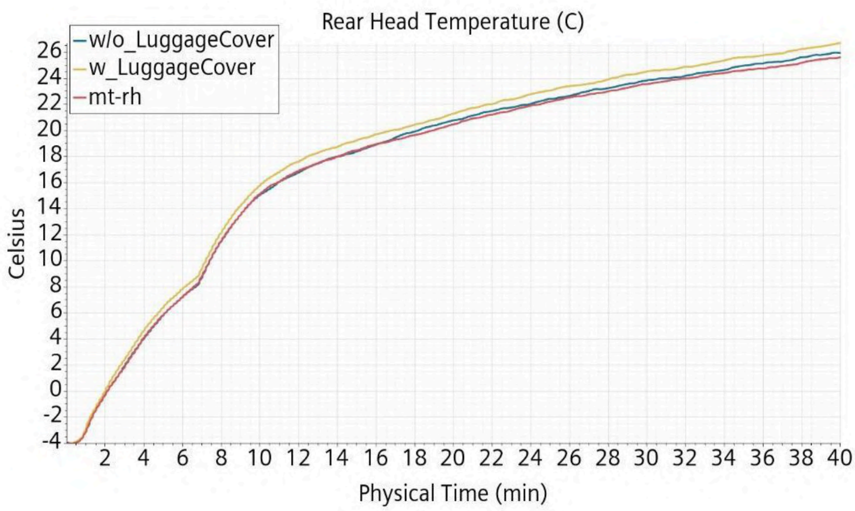
Case2 – The position of outlet is moved to the sides as it is in xc40 vehicle

Case3 - Luggage cover is removed

Case4 – the trunk is morphed from 10cm to 75cm (volume increased by 10 percent)

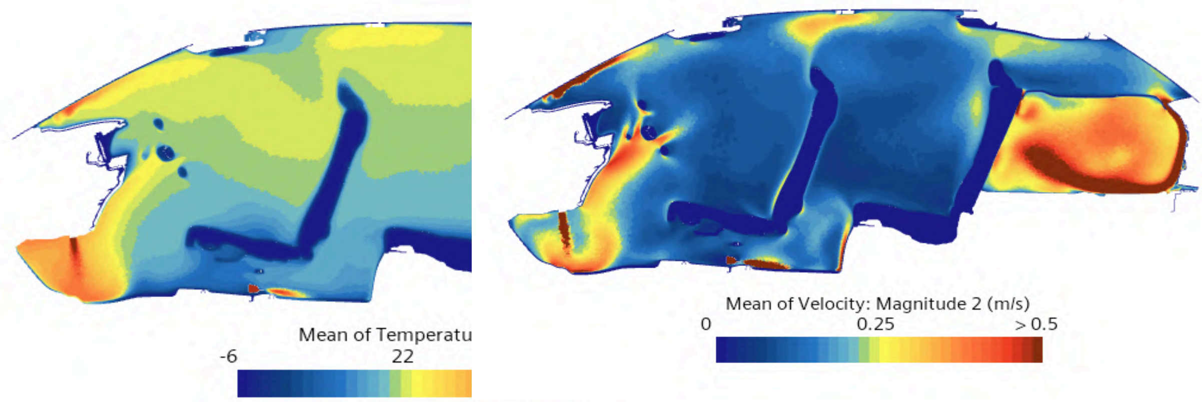




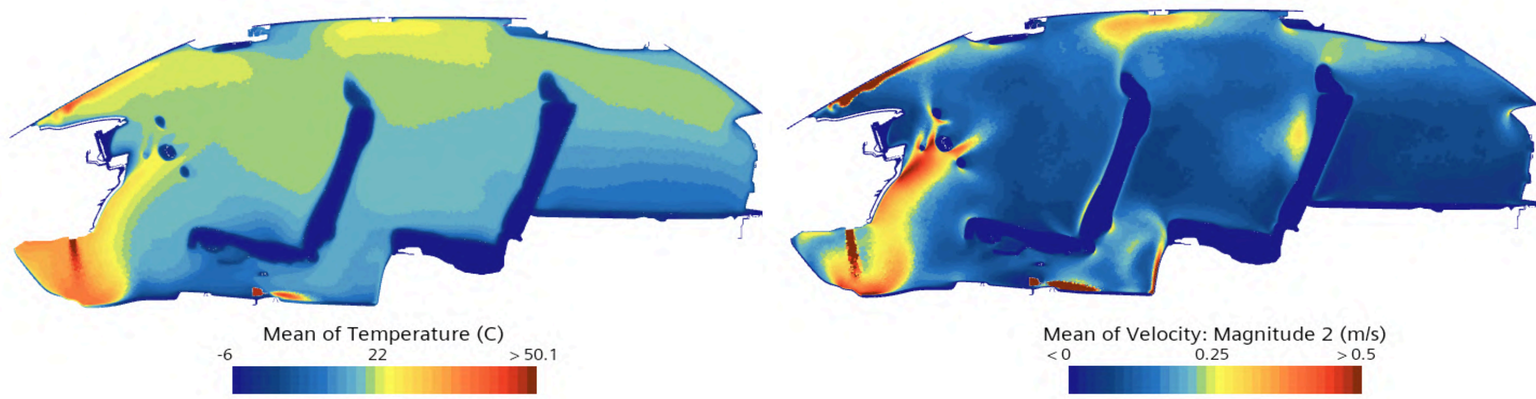


Comparing the plot of all the three cases, we see that case2 is the best model. We don't see any big reduction of heat from cabin to the trunk because of luggage cover. Case 3(one without luggage cover) we see further reduction in temperature, because some of the heat from the cabin is going to the trunk. But we don't see a big difference between morphed trunk and case 3(without luggage cover). They have a very small difference.

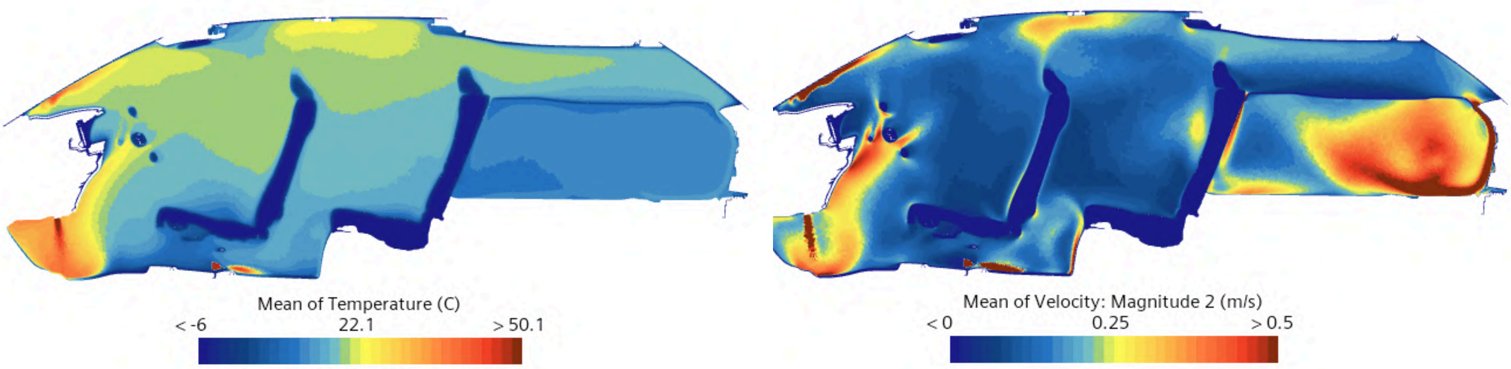
withLuggageCover



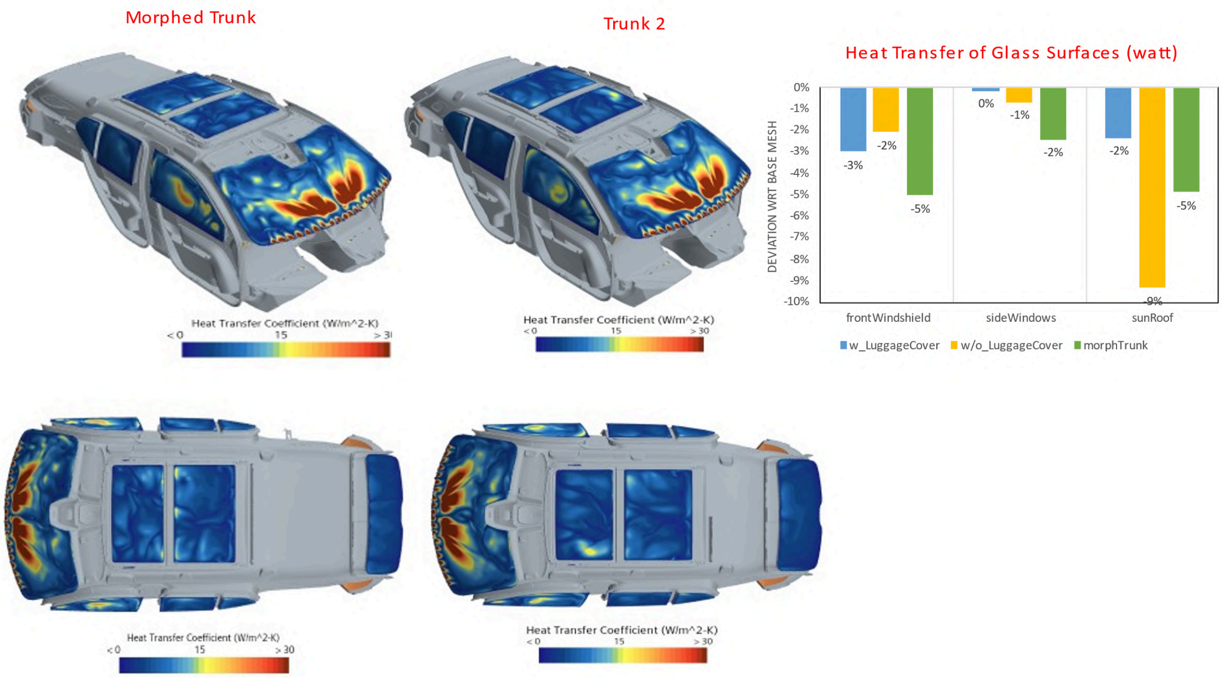
Without LuggageCover



Morphed Trunk



If we look at the temperature profile, we see that case 2 we add the trunk area to the cabin but we don't see a big amount of heat loss to the cabin as we have luggage cover that prevents the higher heat loss. In second case (case without luggage cover), we see lower heat around the mid and foot position compared to case 2. And also more heat is going to the trunk (drawback of removing the luggage cover). In morphed trunk we were supposed to see a lower temperature but that's prevented by using a luggage cover. We can conclude from it that even increasing the trunk wouldn't affect the results a lot if we have a luggage cover. Since we have a small opening between luggage cover and trunk so we have low heat loss.



Same goes for the heat transfer, case 3 performs the best when it comes to front windshield.Least amount of heat is being lost through front windshield. In side windows and sunroof, case2 (with luggage cover) is the best followed by case 3(without luggage cover) and case 4 (morphed trunk). At sunroof, we are seeing higher heat loss in case 3(without luggage cover)

Conclusion

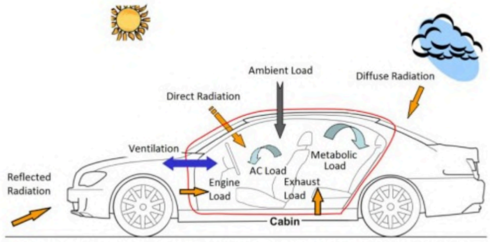


Figure 1. Schematic representation of thermal loads in a typical vehicle cabin [10].

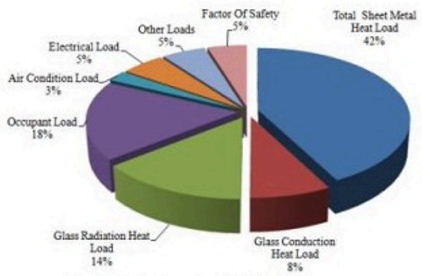


Fig. 2. Solar Heat Load Distribution

$Q_{TOTAL} = Q_{metabolic} + Q_{radiation\ (direct, diffusive,\ reflected)} + Q_{engine} + Q_{exhaust} + Q_{sheetmetals} + Q_{glasssurfaces} + Q_{ventilation} + Q_{ac}$

↓

Calculate the energy consumption of each trunk case

↓

Convert it into monetory cost to know how much we can save

The last task was to calculate the amount of heat loss in each case and then calculate the energy consumption. As in cold weather, radiation has very little impact so we neglect the effect of radiation.

Metabolic Load QMet = Sum of (M x BSA), Where M is the passenger metabolic heat production rate.For a driver and a sitting passenger, the values can be estimated as 85 w/m²and 55 w/m² (ISO 8996 Ergonomics of the thermal environment Determination of metabolic rate) respectively. BSA (body surface area) = 0.202 x W 0.425 x H 0.725 Where W and H are the passenger weight and height respectively. Average BSA is 1.73 m2. For a single person assuming average heat load = 116 W (as per ASHRAE)

Heat Load due to Sheet Metal = Us*As*(ΔT)s

Heat Load due to Glasses = Ug*Ag*(ΔT)g + (diffused radiation)*Ag*τ + (direct radiation)*cos(i)*Ag*τ

Air Infiltration Load: Amount of air leaked = ½ V m3/hr; (V = vehicle speed in km/h), Assuming 50 kmph, air volume leaked = 25 m3/hr. Air Infiltration Load = 25 x Air density x (Enthalpy of Infiltered Air -Enthalpy of Cabin Air)

Heat generated by blower motor = Blower Wattage x Load factor / Efficiency

We calculate only the metabolic, sheet metals of cabin, glass surfaces ignoring all the other variables in the formula. Based on that we calculate the total heat transfer, we convert the wattage into kilowatt and divide it by hr to calculate the unit load. Then we multiply the unit load with the price of 1 unit hence gets the monetory value of each case.

Scripts /Files

In the following location

/vcc/cae/nobackup/cfdcar/thermo/internship/IAESTE/Hussain/Reference_case/STARCCM_finishedSim>

You will find all the completed simulations used for analysis. Two important folder are corrMesh and MeshSettings folder. Each folder represents the following information

CorrMesh > contains all the timestep, iterations, simulations of the improved mesh

Anandh> contains all the simulations related to Anandh´s Mesh

Mesh settings > contains all the files in which mesh setting was changed.

Scripts

corrMesh_case12 is the script used for all the files in which mesh settings are changed.

corrMesh_2t40 and corrMesh_4t5i are used for individual simulations of 40 iterations and 5 iterations

Trunk1 and Trunk2 are simulations used for both cases of trunk

cabinHeatup_case12 is the script being used for Anandh simulations

Excel

If you want to understand all the graphs shared here or in presentation in more details, refer to the excel file by the name RESULTS. It contain all the data. Important sheets to consider are

- corrMeshComparison – contains all the data from corrMesh
- StatisticalReports – it involves comparison of different delta time vs monitor volumes
- MeshSettings – information regarding different mesh settings