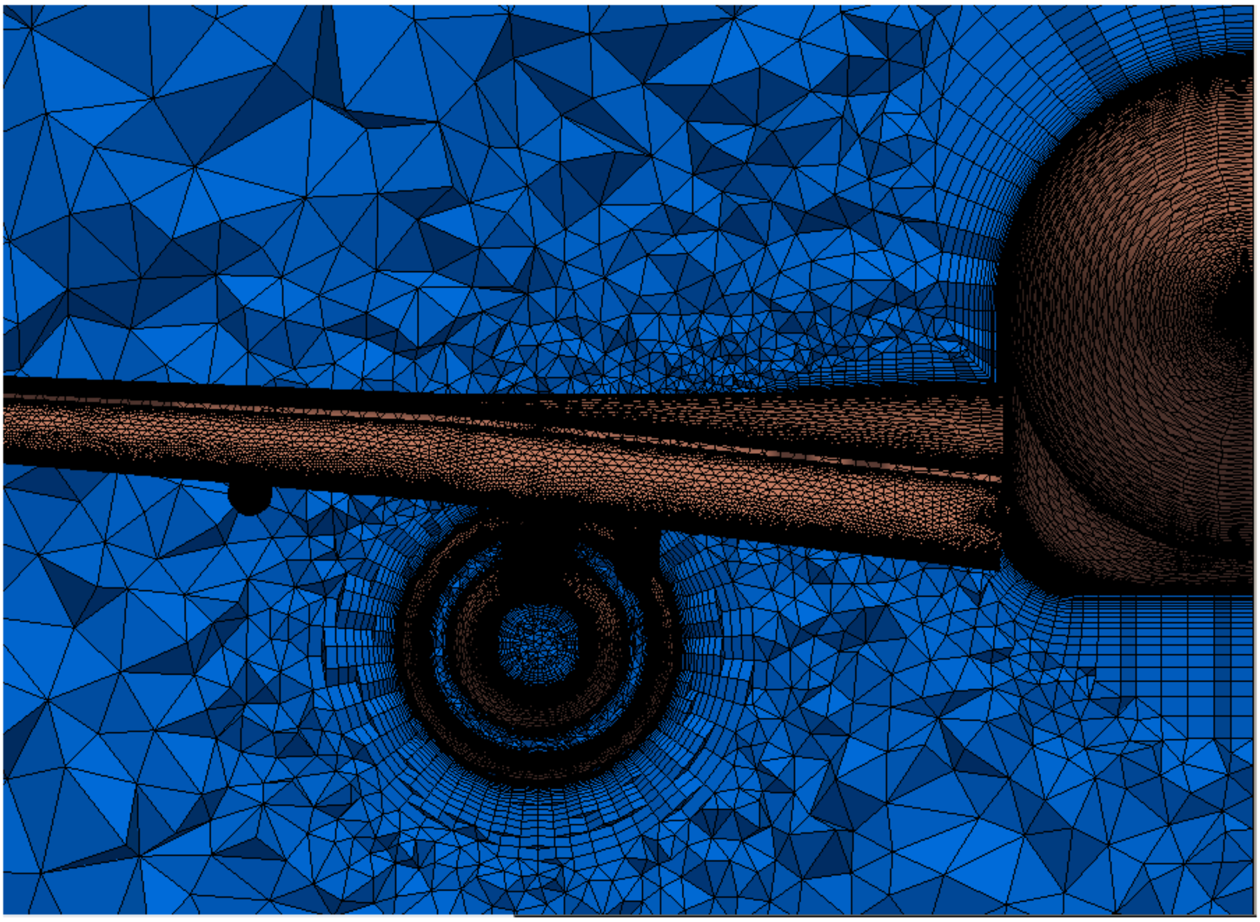


Ennova Tutorials

User guide
2019

Table of contents

Tutorials

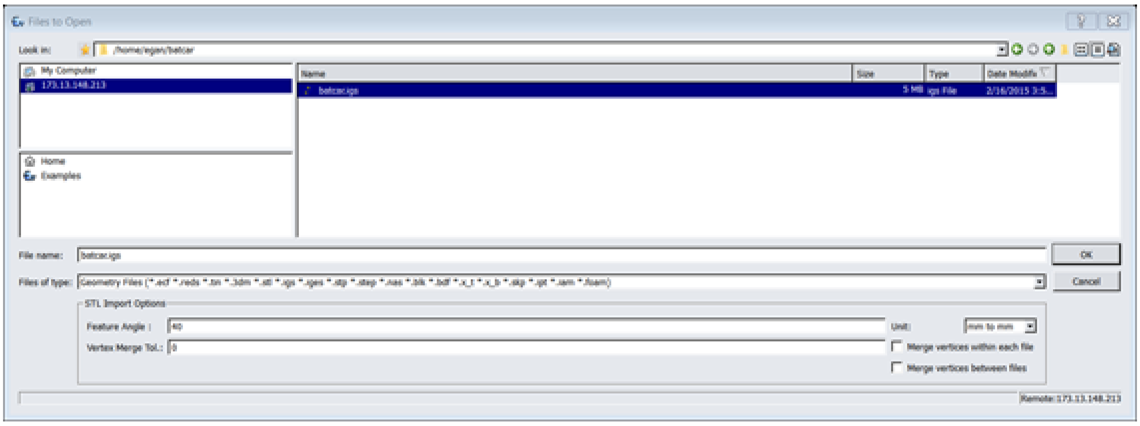
1) Simple Concept Car Tutorial
2) Drivair Car Tutotial
3) IconCFD Tutorials (*These tutorials are strictly for using Ennova with IconCFD*)
    - Particle Simulation
    - Congugate Heat Transfer
    - Shrinkwrap

Bat Car Tutorial

Open file in Ennova

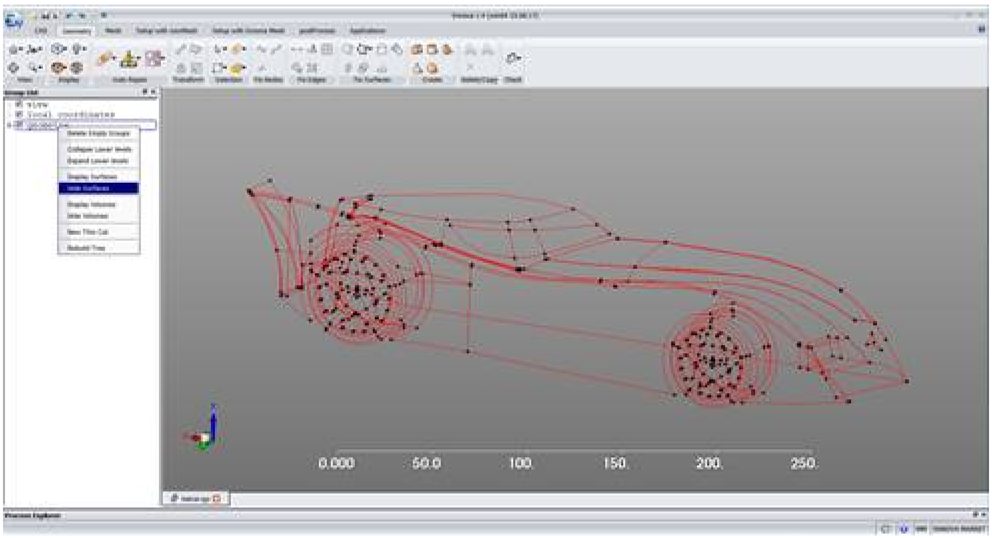
Make a new directory BATCAR on your host/server computer in your account. From the Ennova Website download the file BATCAR.igs to this directory.

If Client/Server is on the same computer, navigate to the directory and open BATCAR.igs. If the Server is remote, choose the IP of that computer from the list and navigate as usual. There is no difference when running Ennova at a user level if the Server is local or remote, except for this step.



BATCAR on loading is all RED edges

Surfaces are not connected. They do not form any closed volume or surface quilt / shell.



Car repair

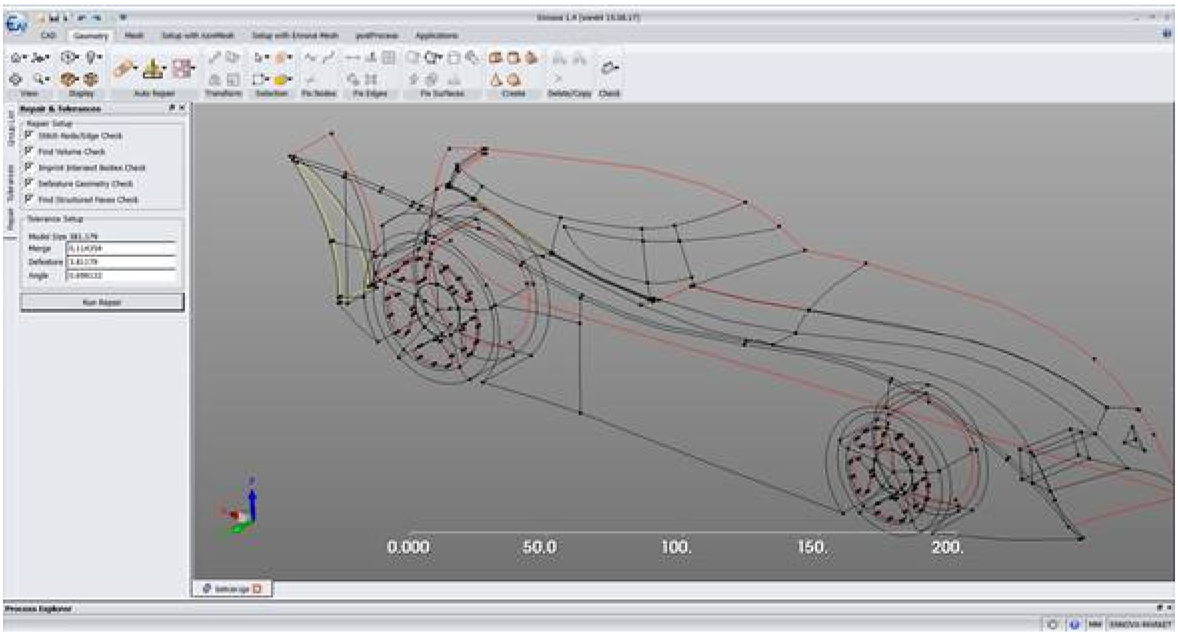
In fact, in addition to the surfaces not being connected, there are not enough surfaces to form a volume. Nor are there any enclosed outer CFD volumes.
To fix this geometry:

- Create an outer CFD Box
- Stitch and Repair the car surfaces
- Add any remaining surfaces that form holes
- Obtain a closed CFD volume

We will start by repairing the geometry and building half the car. Use the Repair icon (Band aid) menu and click on **Set Geometry and Repair Tolerances**.

Set the Tolerances as follows:

- Merge Tolerance 0.11
- Defeature Tolerance 0.00 (here we are effectively turning off defeature)
- Angle Tolerance 0.784

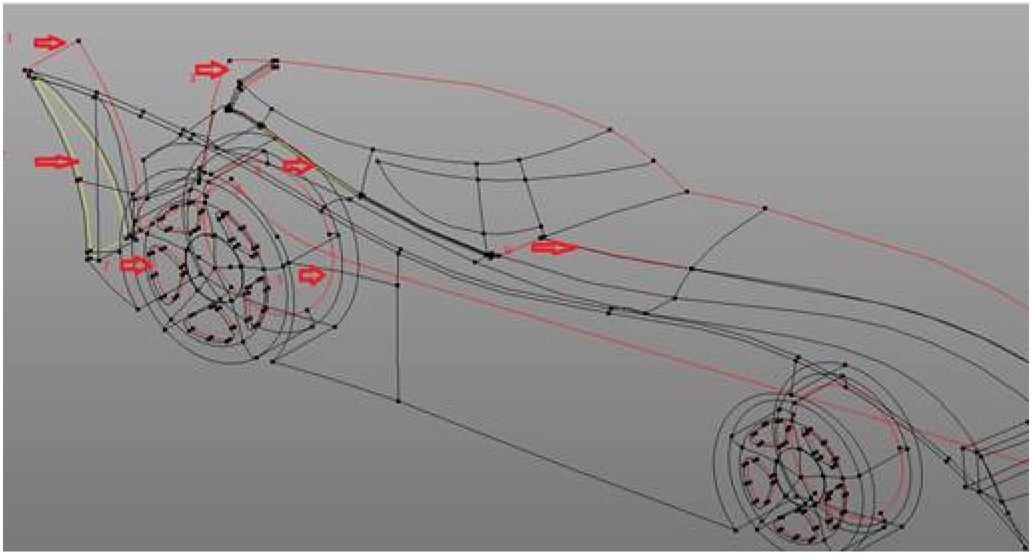


Automatic Repair – Initial Results

After Automatic Repair we find 8 areas where Ennova has not 100% repaired. The goal was to make the entire model BLACK (except a RED single edge around where the symmetry plane will be).

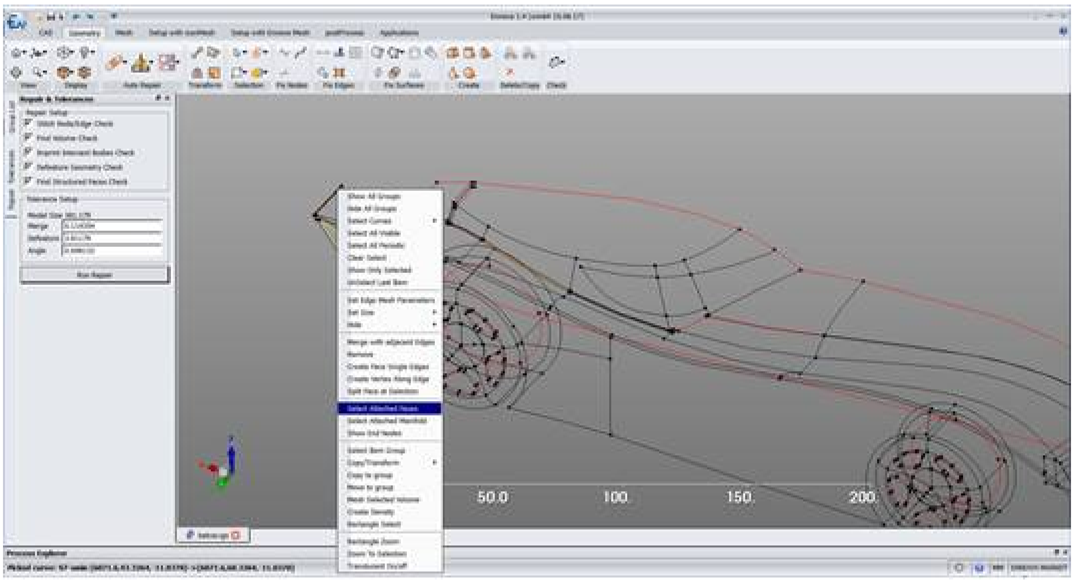
You may have noticed that we set the defeature tolerance to zero. Typically there will be features in the CAD that are not important to CFD. These features can be removed later, but once removed may be difficult to bring back if needed. ***Our suggested approach is to defeature as a last step***. Please note that if applying a very fine boundary layer mesh, there are some small fillets that may be important to the boundary layer flow.

8 different initial results are left to resolve.

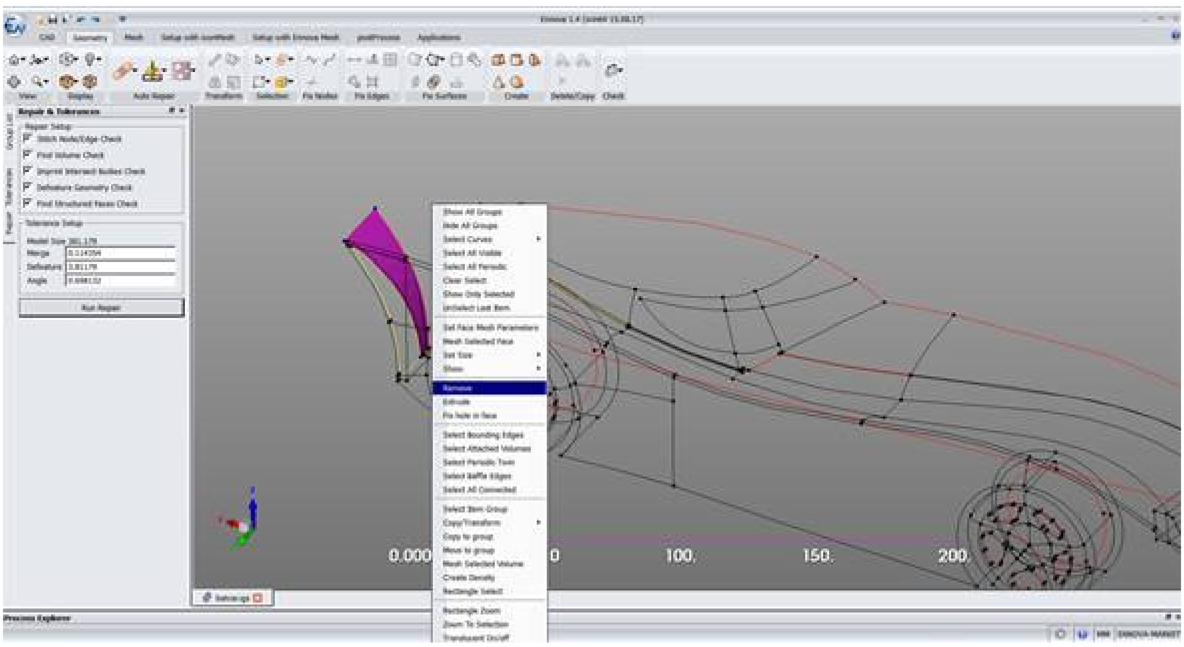


Resolving Initial Results – Remedy 1
Result: Extra Surfaces Require Removal

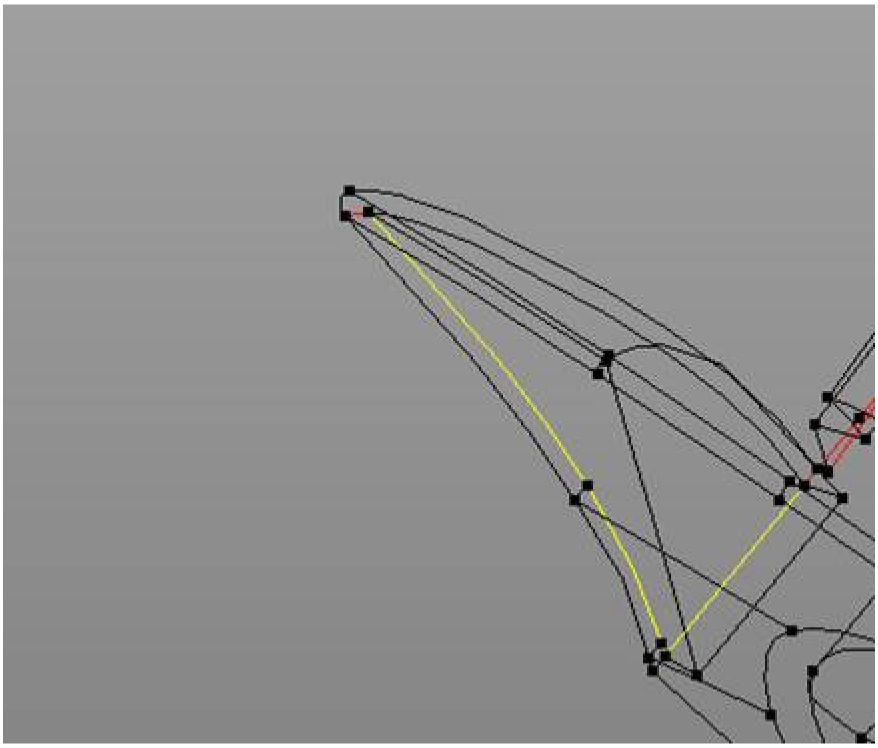
Select the top RED edge with RMB, and **Select Attached Faces**.



Then **Remove** the selected face again with RMB and the initial result is resolved.
Extra RED edges disappear and the model is watertight and BLACK.

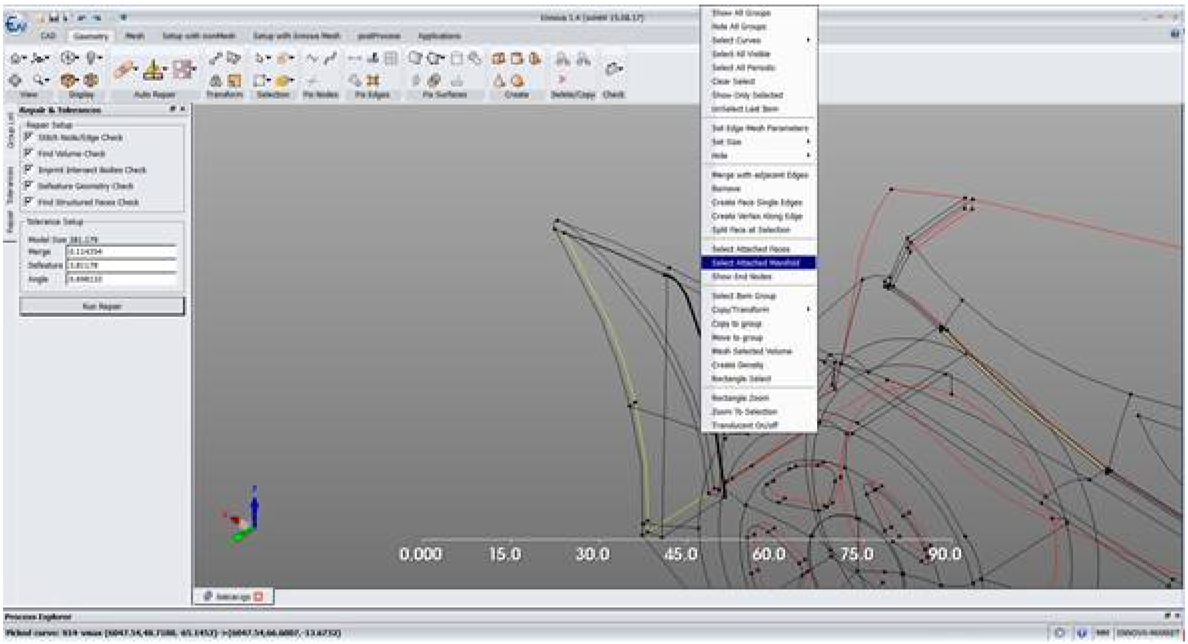


If we had selected a BLACK edge, then two of the faces would have been selected/highlighted. These selected faces are not displaced.
Use **RMB -> Show Items** to display them.

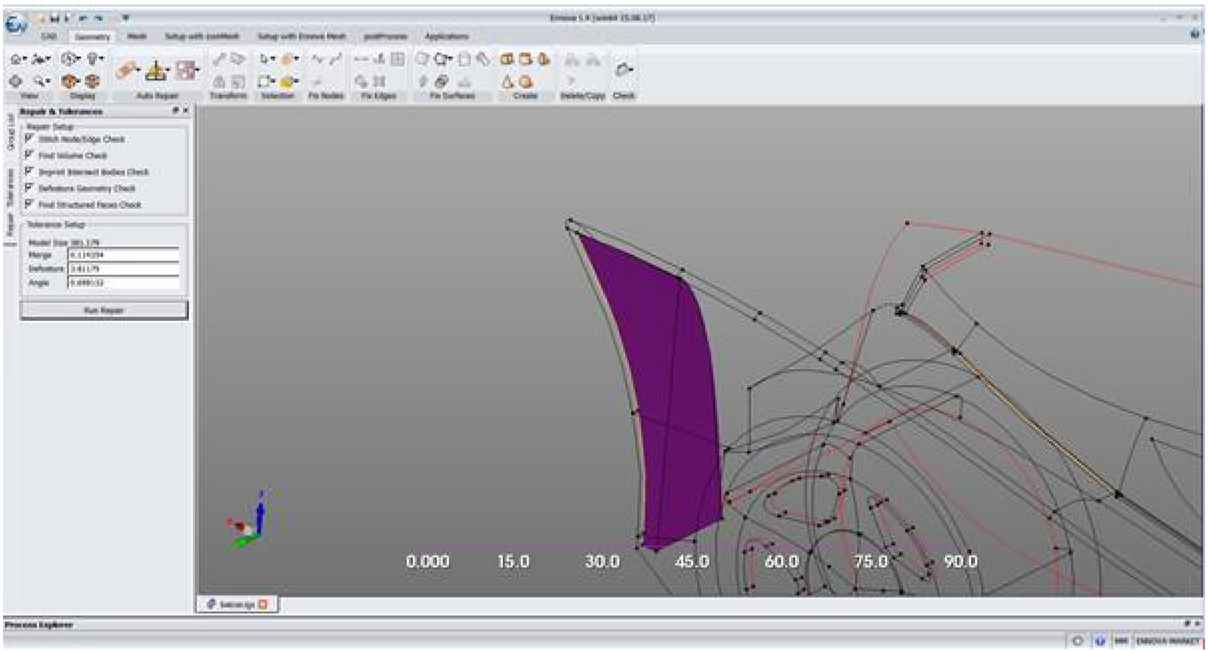


Resolving Initial Results – Remedy 2
Result: Tail Surfaces - Extra and Missing

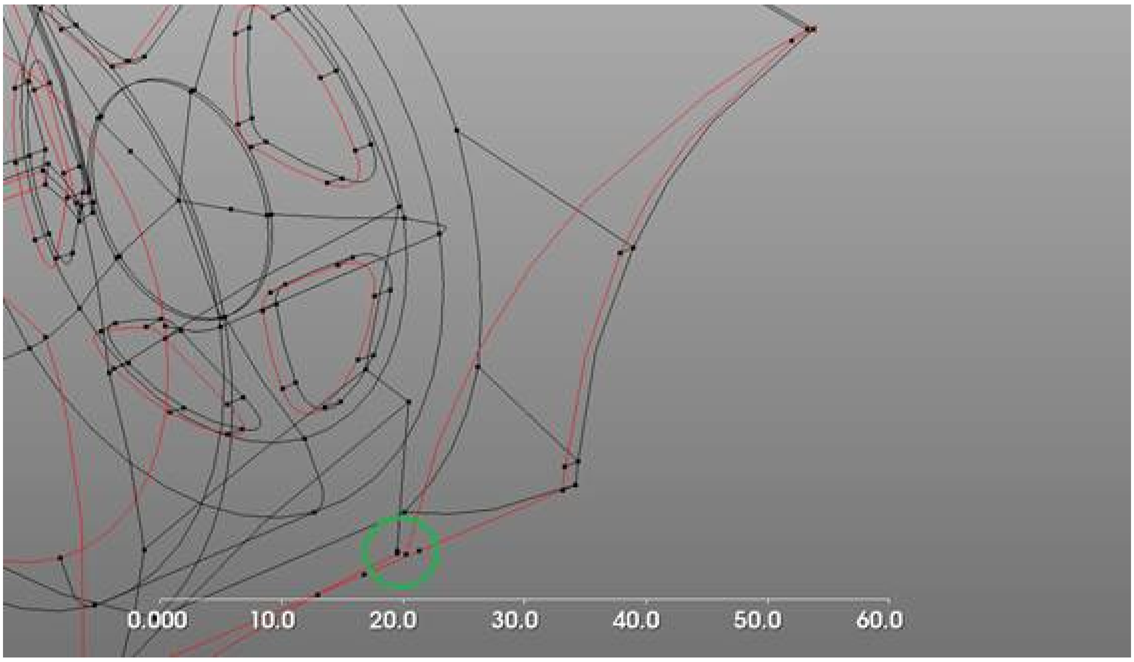
There is an extra volume in the tail fin area of the car marked by the YELLOW edges. Ennova can pick this entire volume minus the shared faces automatically. Use **RMB -> Select Attached Manifold**. Select one of the BLACK interior edges.

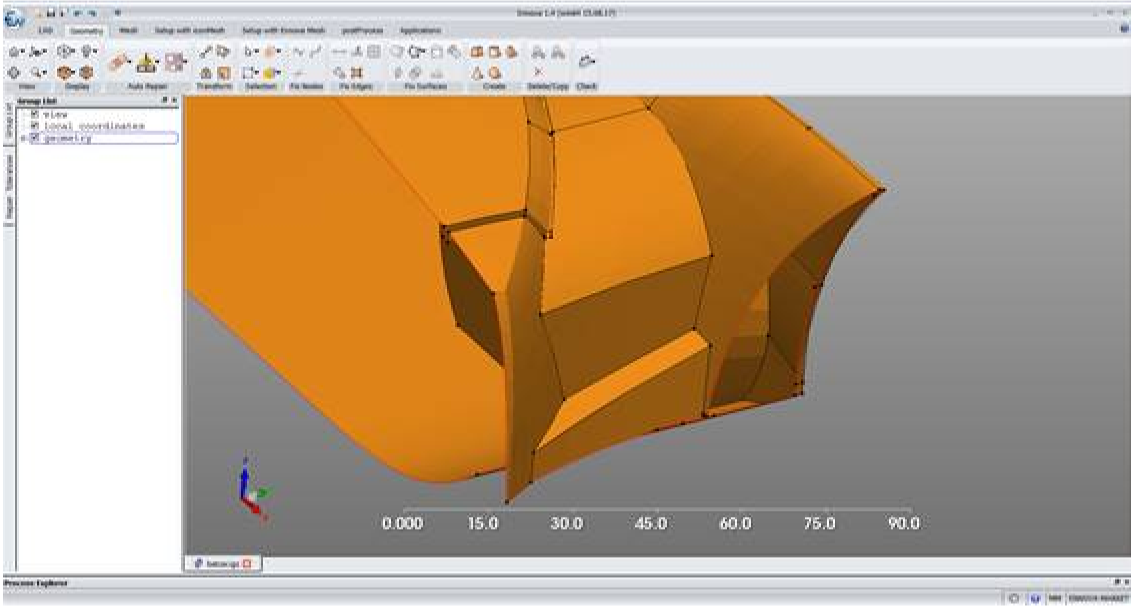


Once the region is selected, use **RMB -> Remove** to remove the extra geometry.



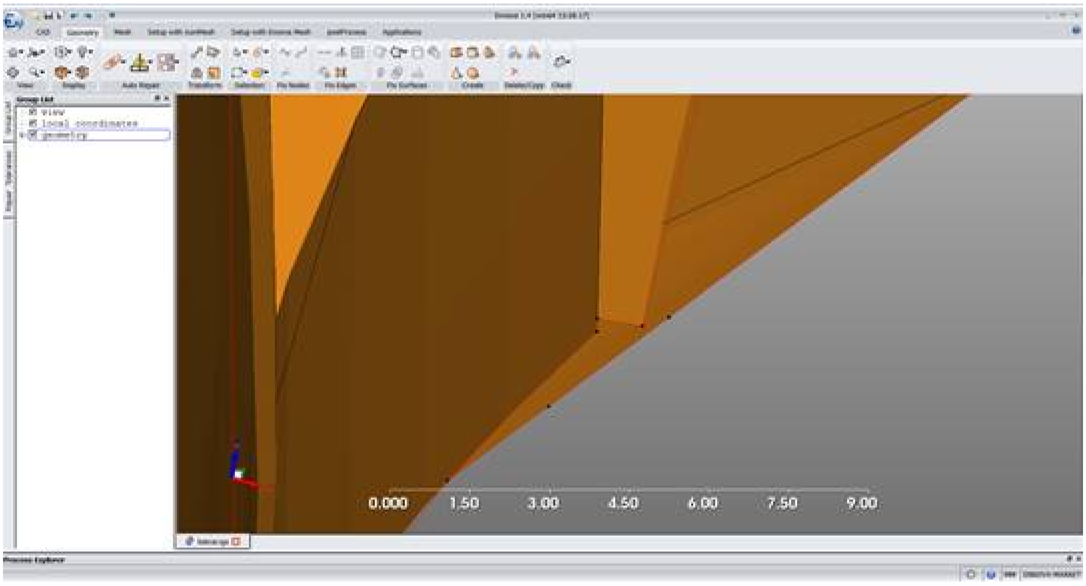
The tail surface is missing, so there are some surfaces in the bottom that should trim to each other but do not.





Next we will look in detail and learn some more Ennova commands.

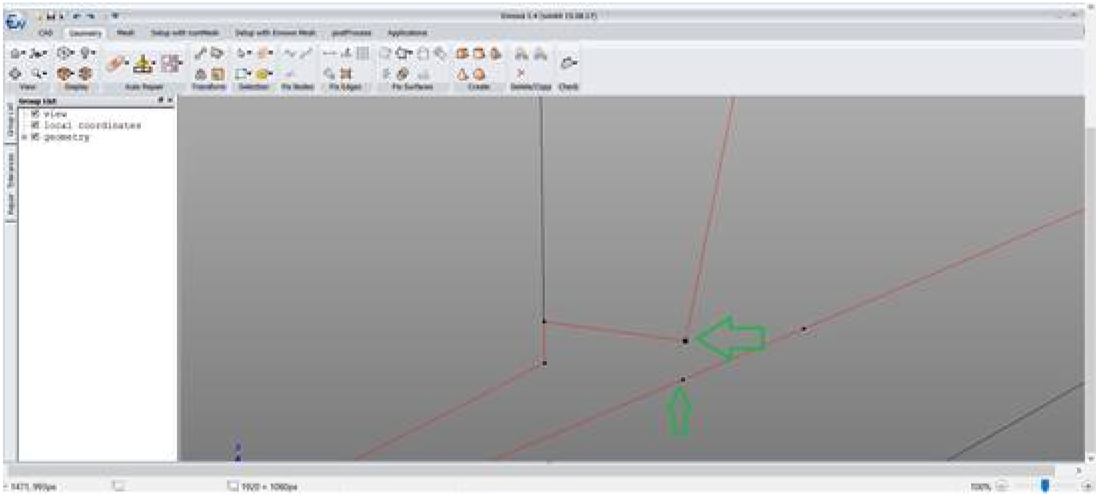
First we will fix the tail surface and then split and connect the floor.



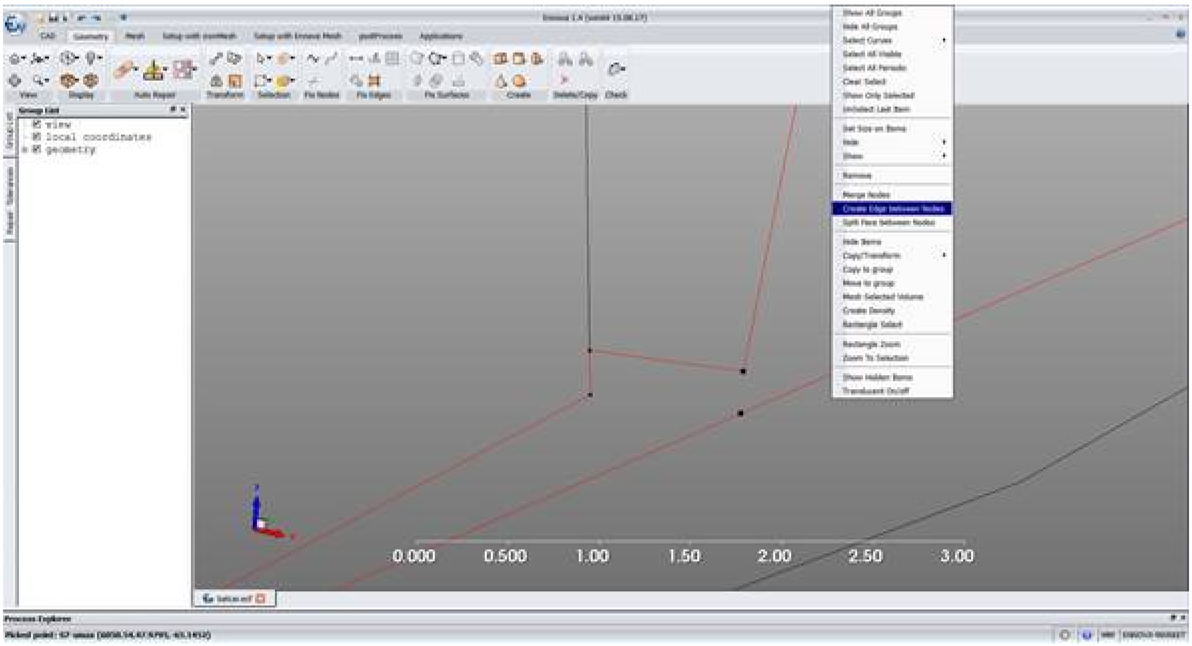
Note: We could have increased the internal tolerances to allow Ennova to do some of this work automatically, but here we want to demonstrate various repair methods.

Concentrating on the area in the circle, we see a surface is missing. First use **RMB -> Project Vertex** on the edge to create a new vertex.

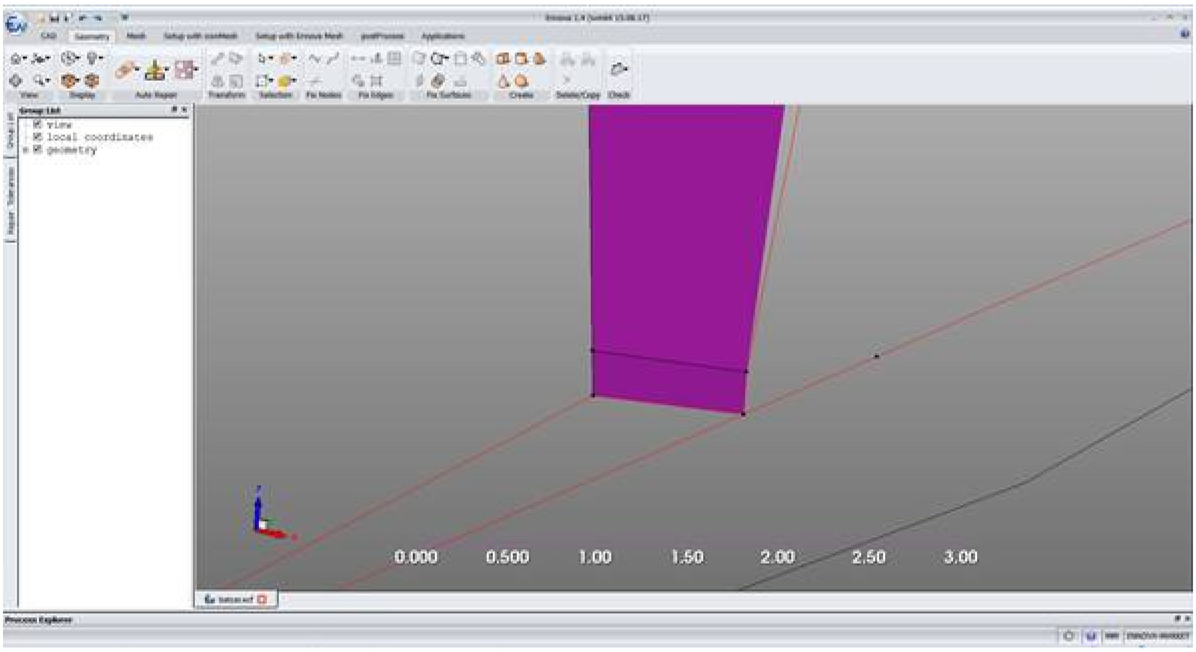
Next, click-select the first vertex, select the second vertex using CTRL and then use **RMB -> Create Edge between Nodes**. Notice that the RMB is context sensitive, so with different items selected there will be different commands available. Do the same for the other nodes of the rectangle.



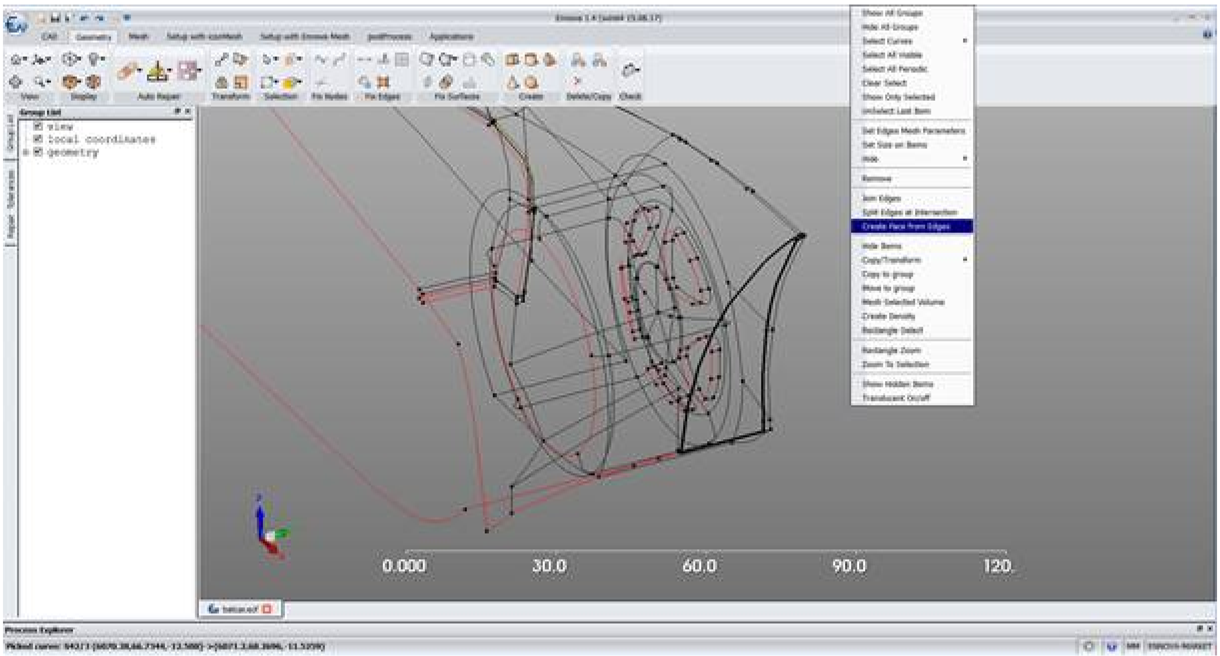
Now there will be 2 RED edges and 2 BLUE edges. The RED are connected to one surface and the BLUE are free. Experiment with **RMB -> Select Attached** to verify.



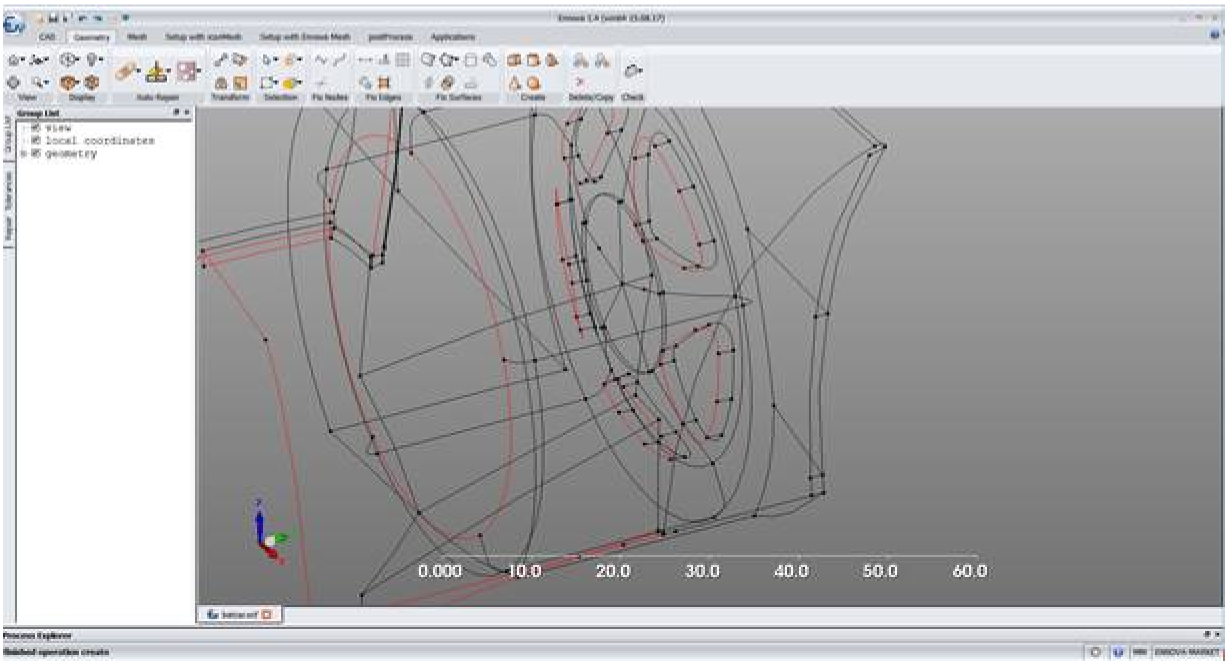
Next, pick the 4 edges and **RMB -> Create Face from Edges** to make a new face. Notice the edges change color to match the new status.



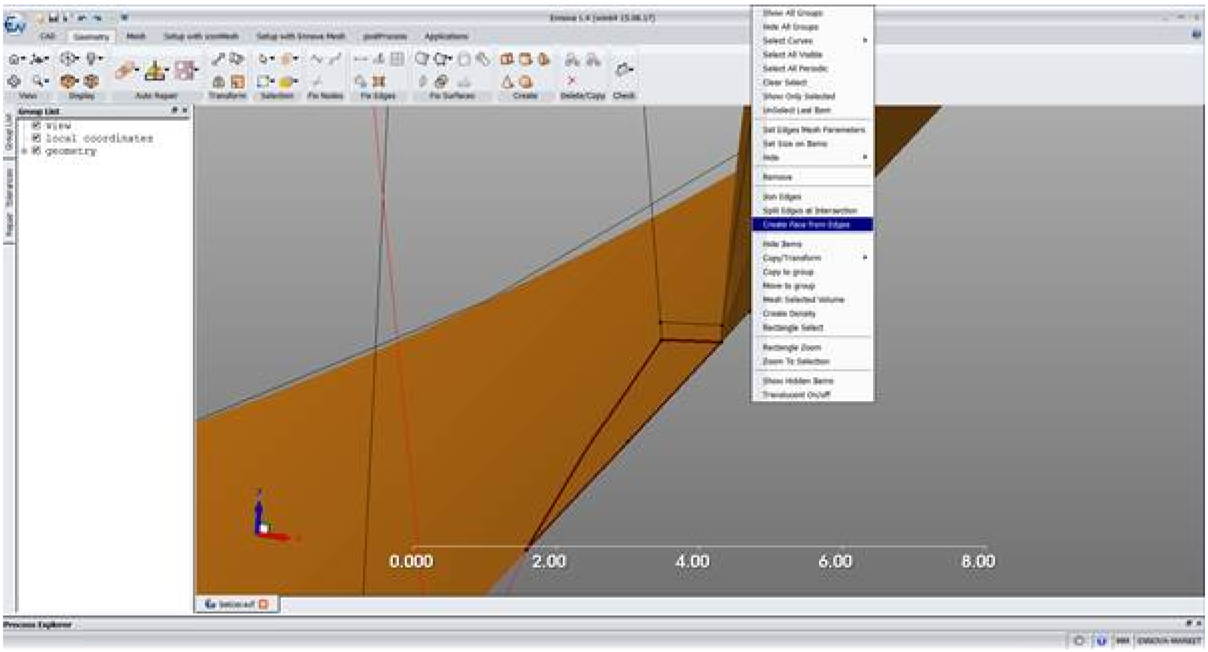
On the fin, select one RED (single) edge and use **RMB -> Create face from single edges**.



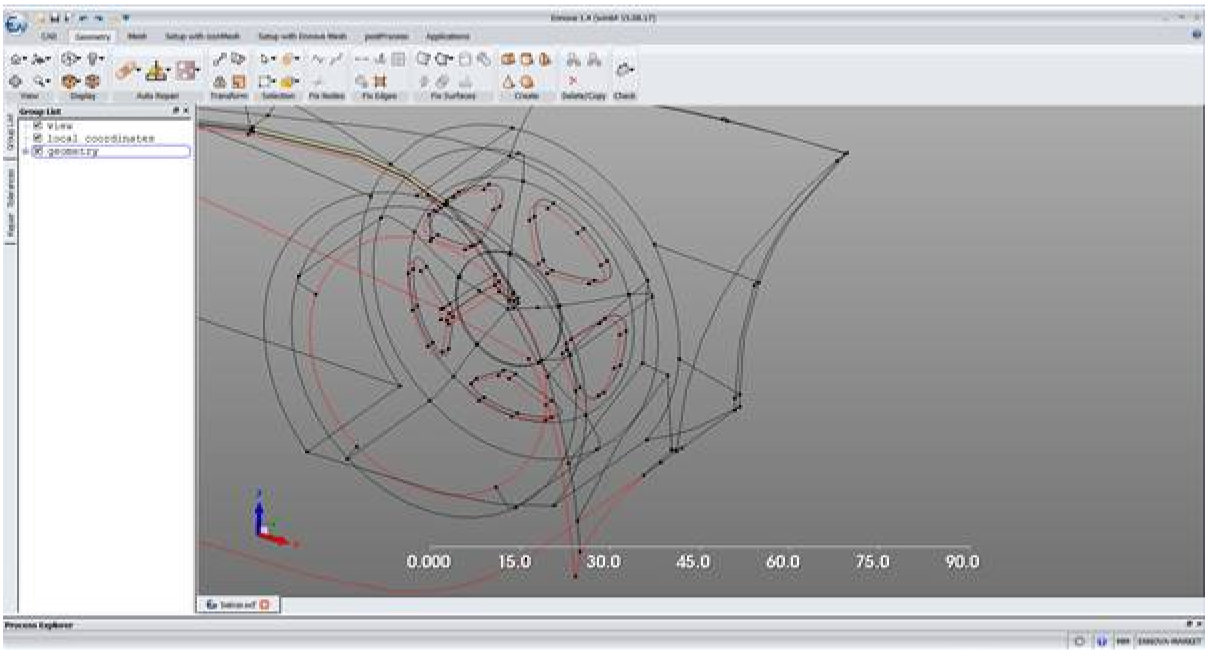
A new face is created to fill the loop and all edges are BLACK.



There is one more missing Face. Select the edges as shown and use **RMB -> Create Face from Edges** again,

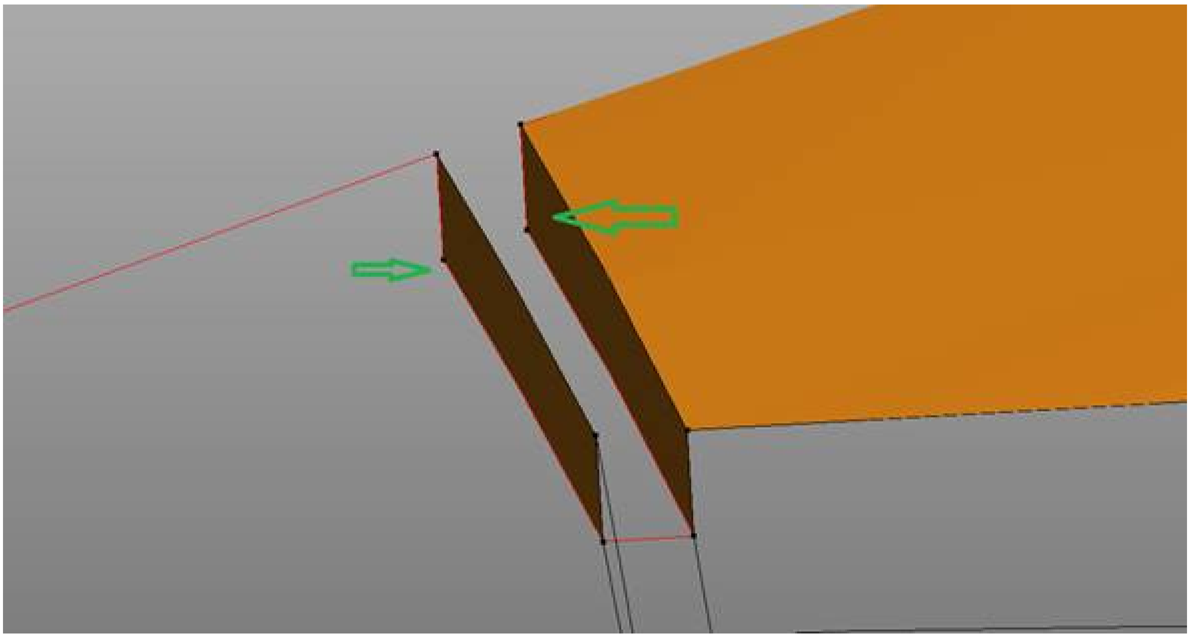


and the Remedy 2 is complete.

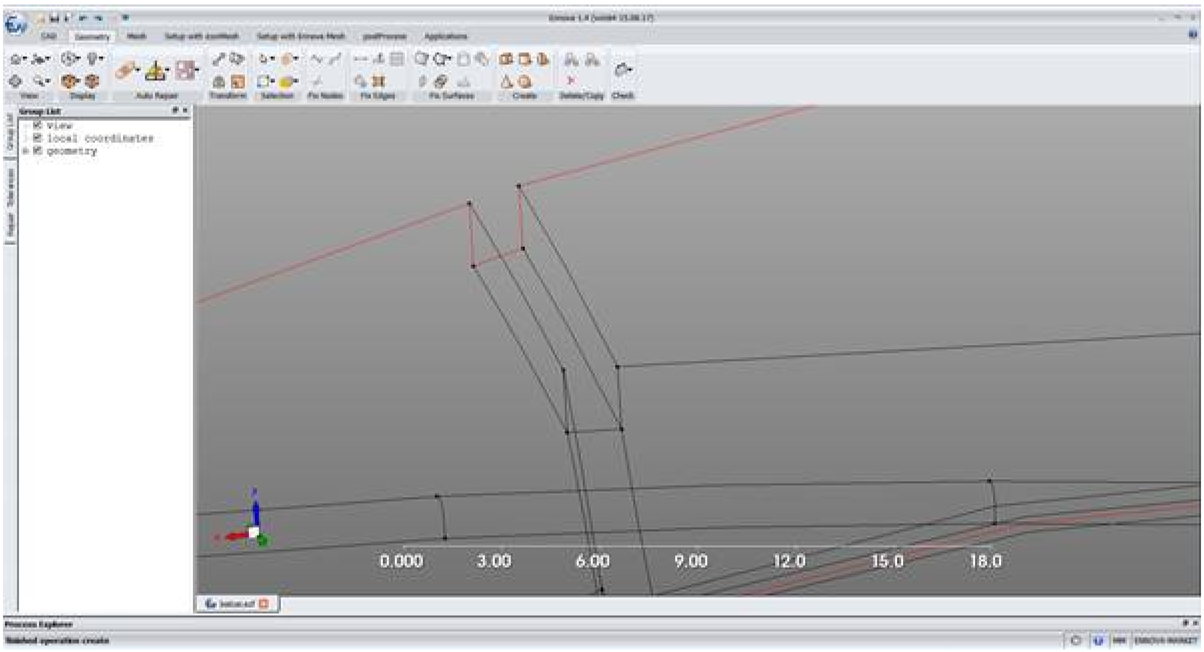


Resolving Initial Results – Remedy 3
Result: Missing Edge and Face

Here we use the exact same methodology as in Remedy 2. We first create an edge between the two nodes,

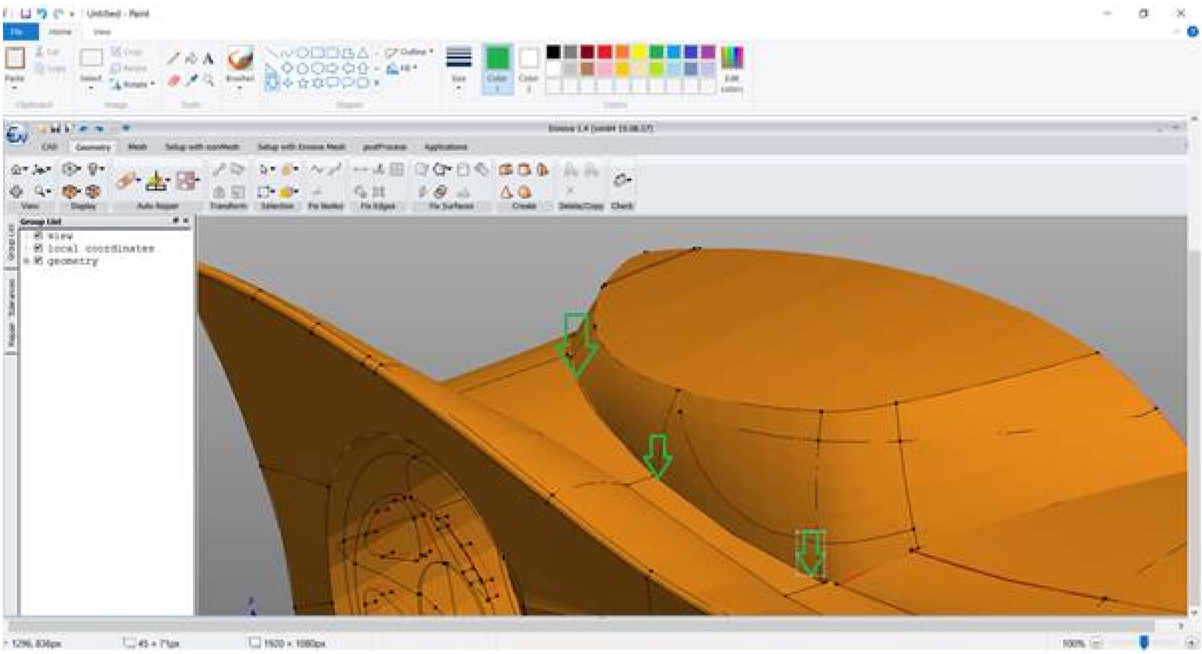


and then select the 4 edges to create a new face.

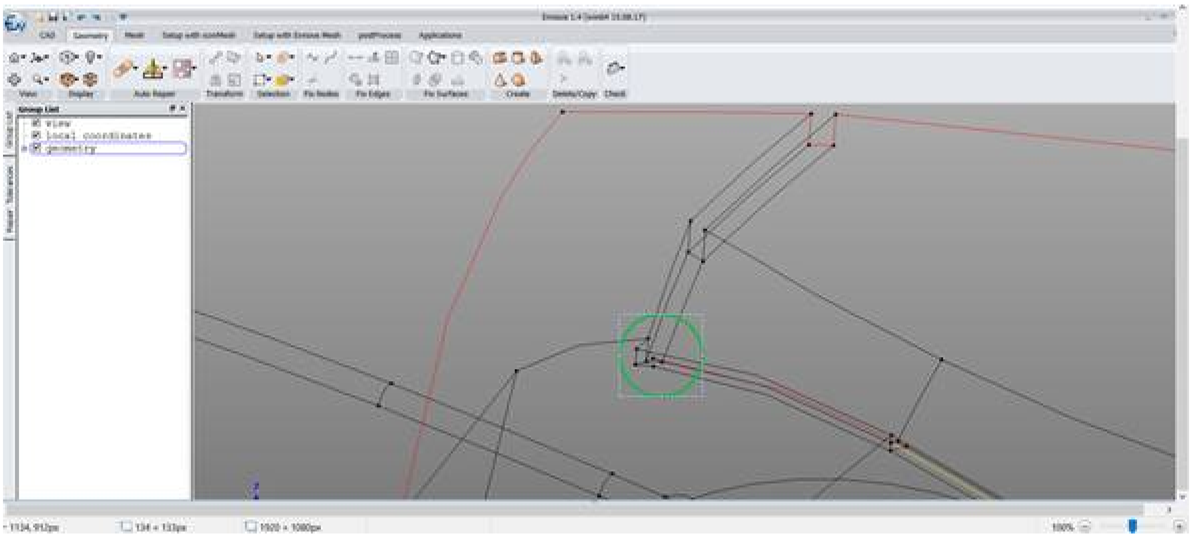


Resolving Initial Results – Remedy 4
Result: Surface Gaps at Edges

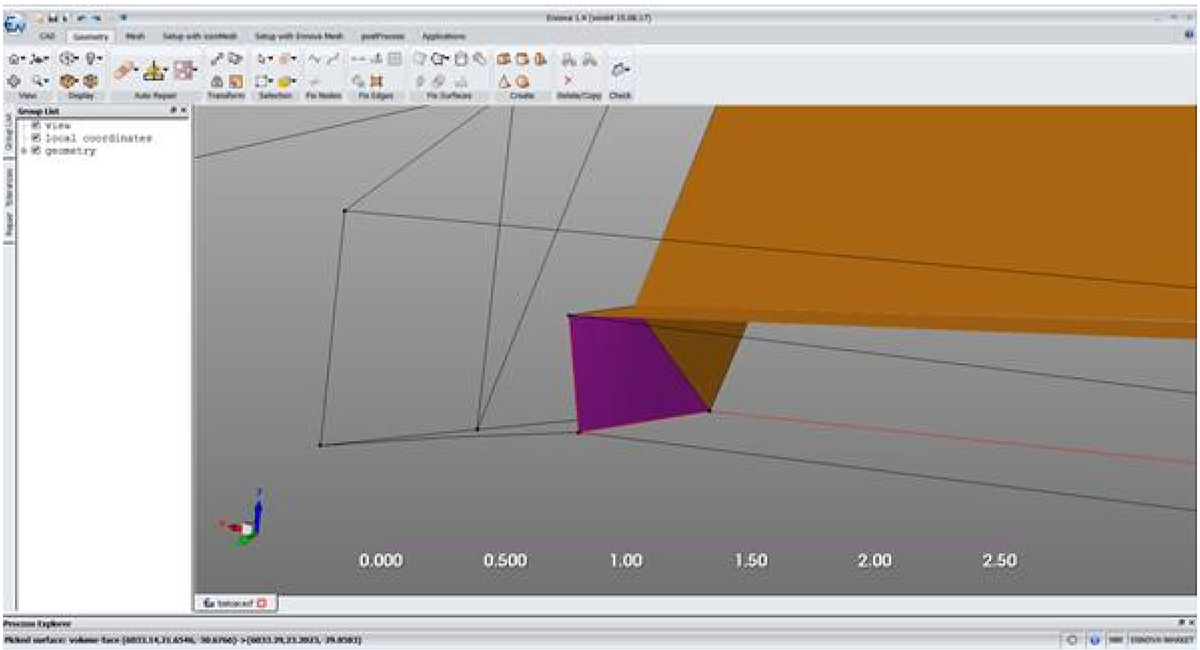
Here the CAD model cockpit does not intersect well with the car body. The surfaces don't quite line up and there are gaps.



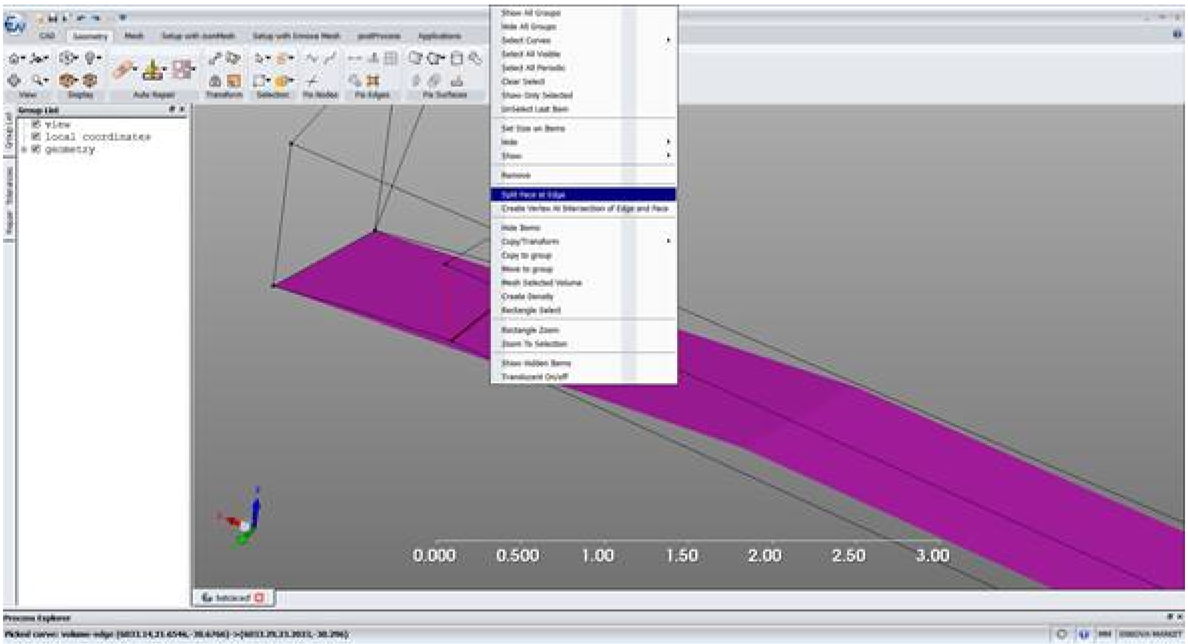
First use your previous skills to build the missing face in the green circle.
Hint: Use **RMB -> Create edges(s)** and **RMB -> Create Face from Edges.**



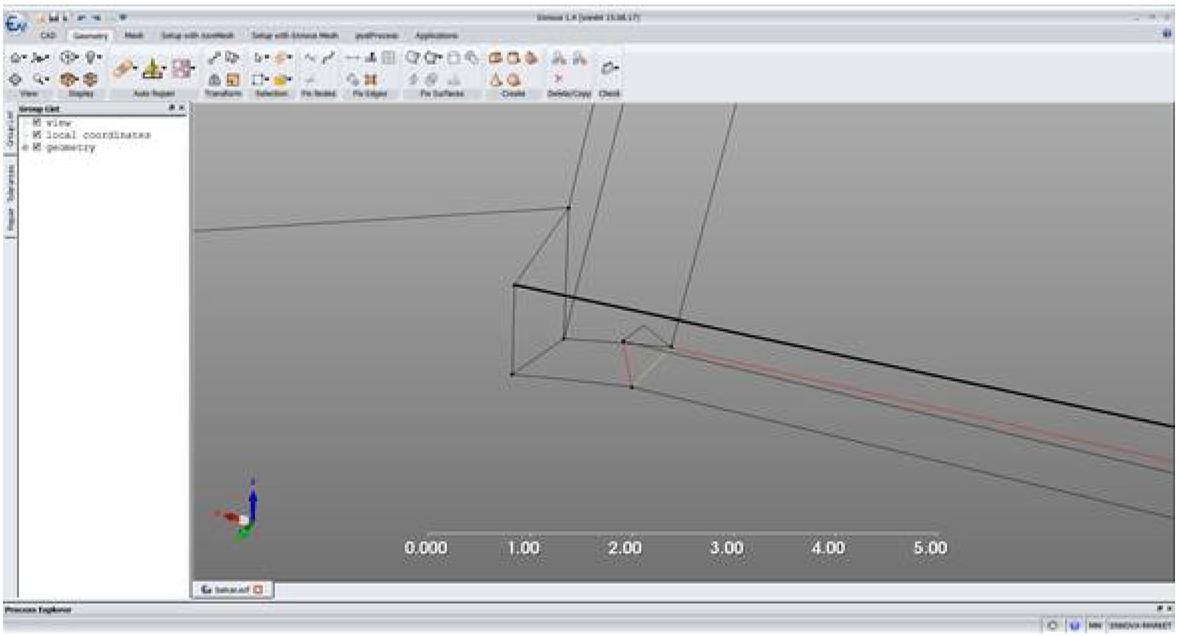
The new face is highlighted.



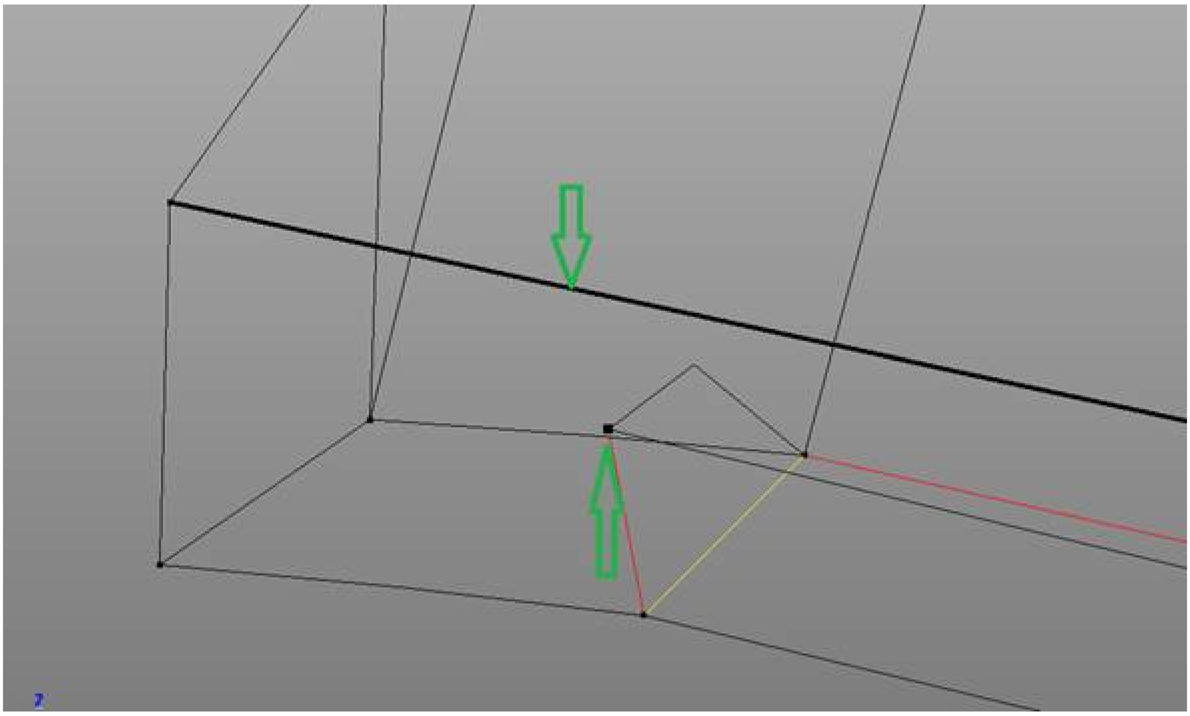
Next, select the face and the new bottom edge of the new face. Use this edge to **RMB -> Split Face at Edge**.



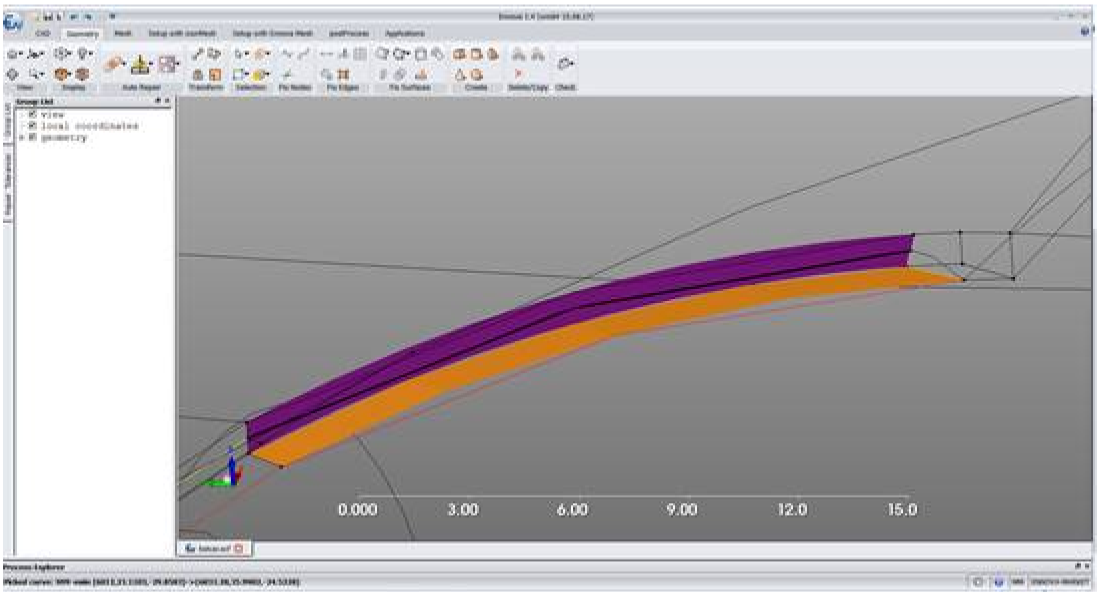
Now Project Vertex to the top edge and create and edge followed by a split using the same sequence as we learned previously.



Then we have the following topology.



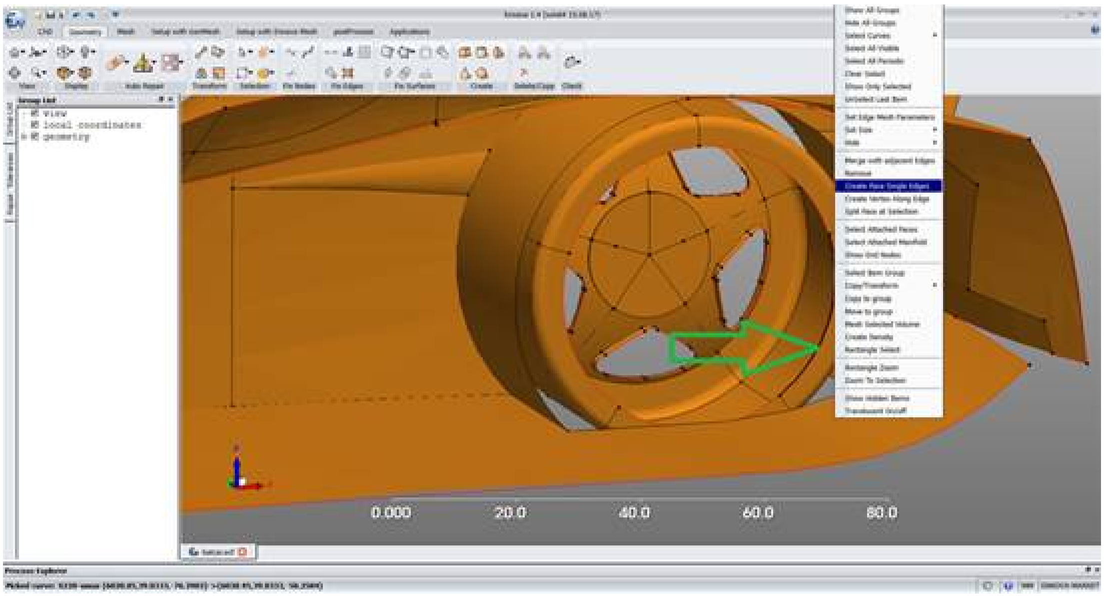
Finally, split the back surface.

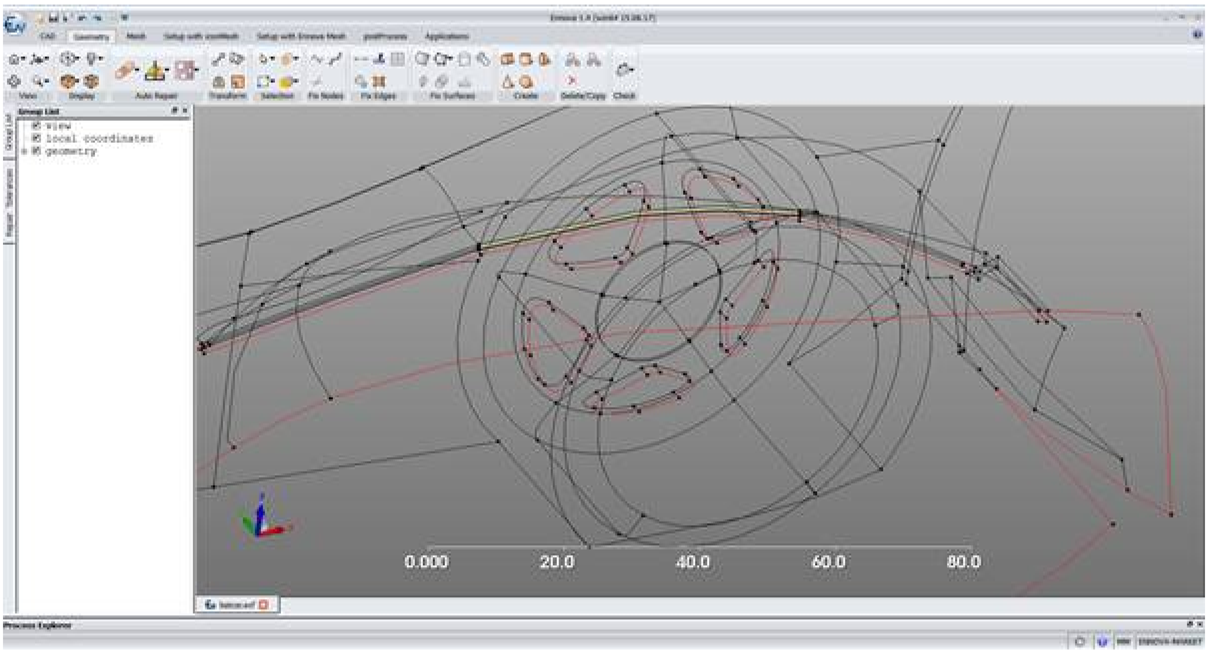


Now the topology is correct at the gutter of the rear of the cockpit. It is not correct at the front, but we will address that in Remedy 6.

Resolving Initial Results – Remedy 5
Result: Surface Gaps on Faces

Here the wheels are open and need the backs filled. Use **RMB -> Create Face from Single Edges**. Note: For simple loops, do not click and choose every edge. Just pick one and Ennova will fill find the loop.

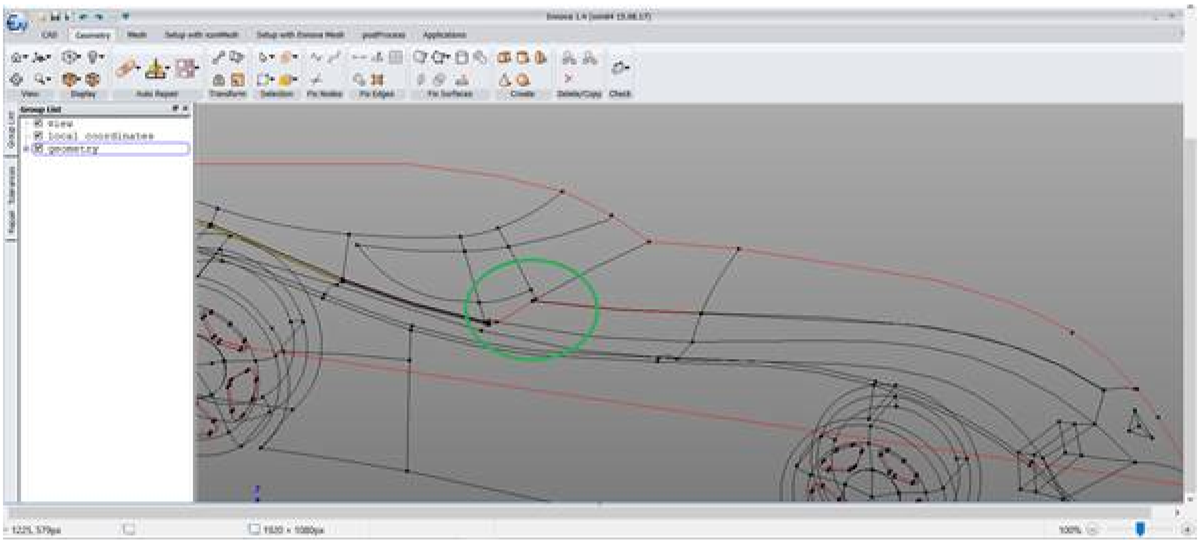




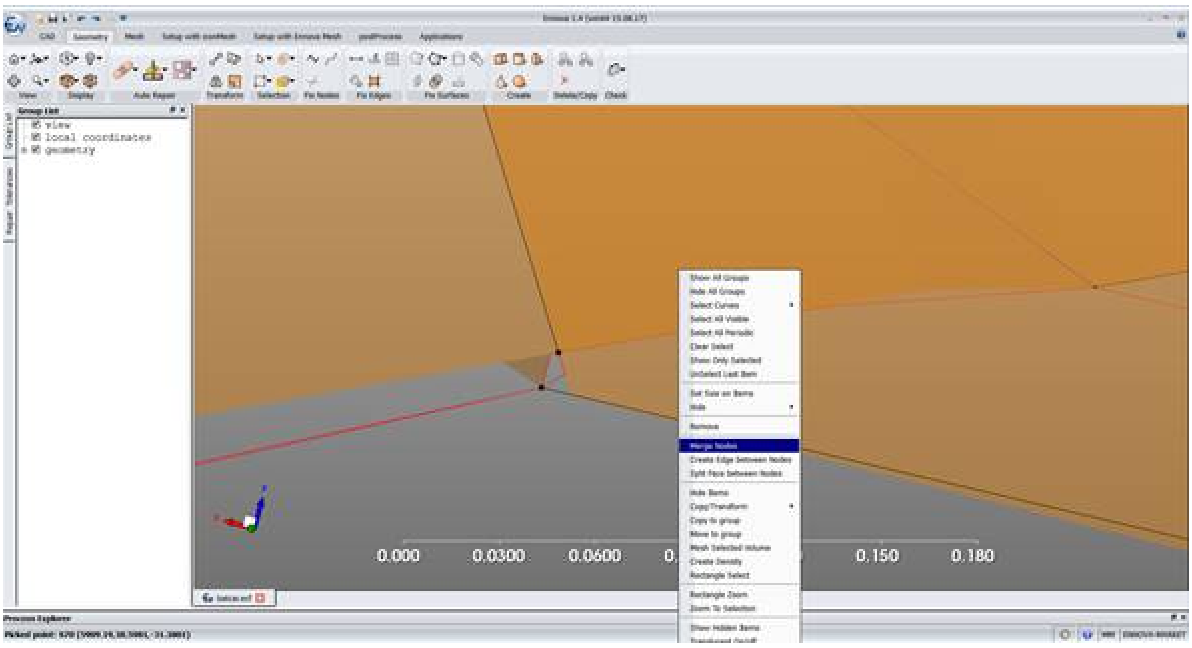
Do the same for the front wheels. Some surfaces are missing as we saw before and some surfaces are too far apart to connect.

Resolving Initial Results – Remedy 6
Result: Disconnected Car Hood and Windshield

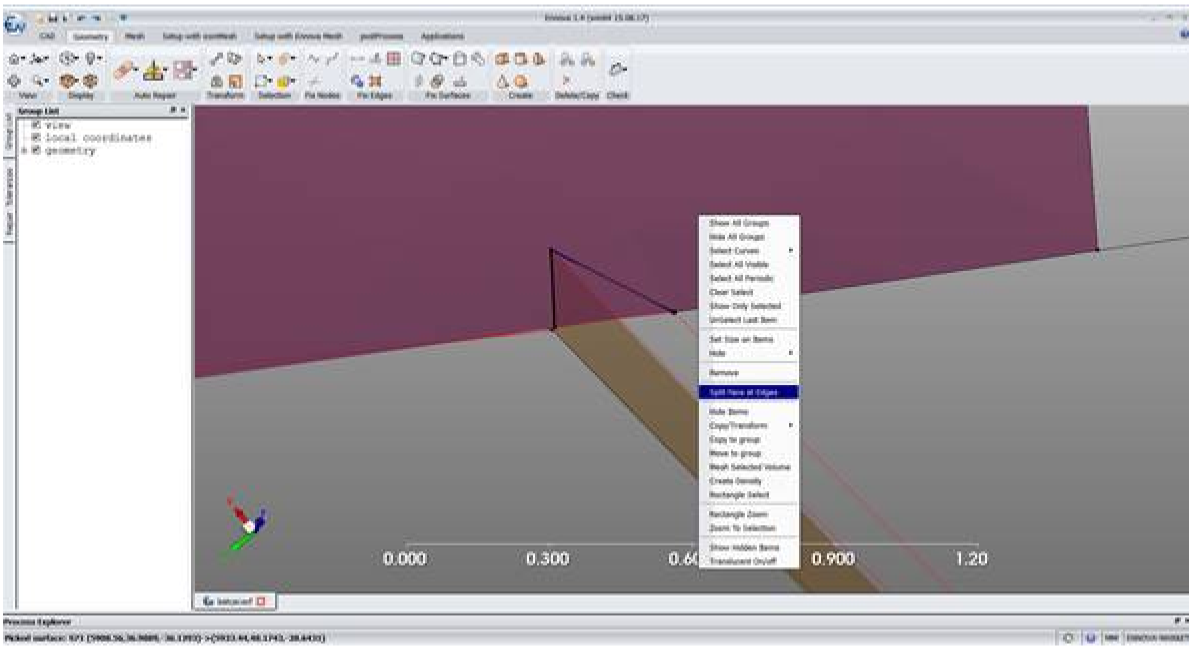
First locate tiny weird geometry at the hood and windshield inside the green circle.



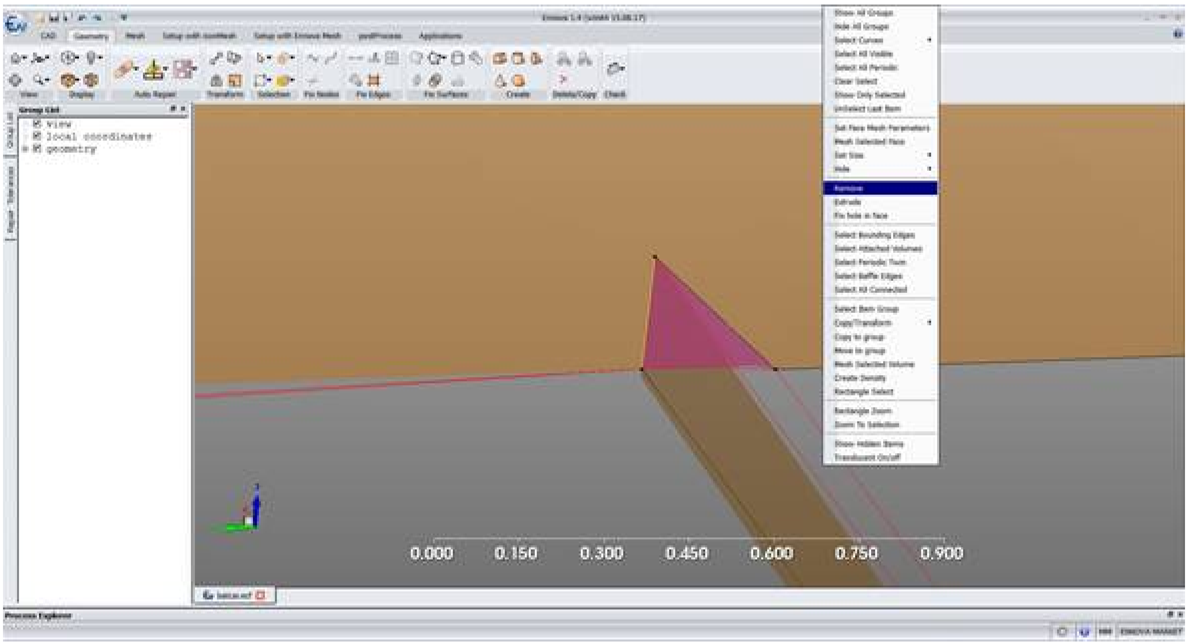
Select nodes and merge them with **RMB -> Merge Nodes.**



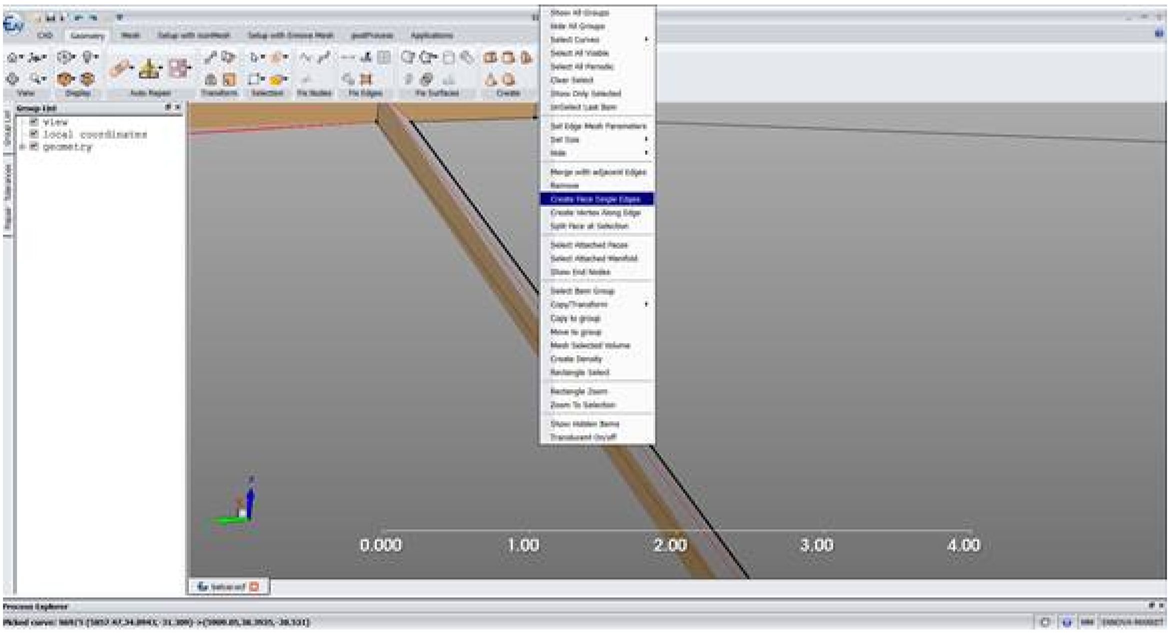
Then use **RMB -> Create an Edge** through nodes that are missing and use **RMB -> Split Face at Edges** to cut the windshield.



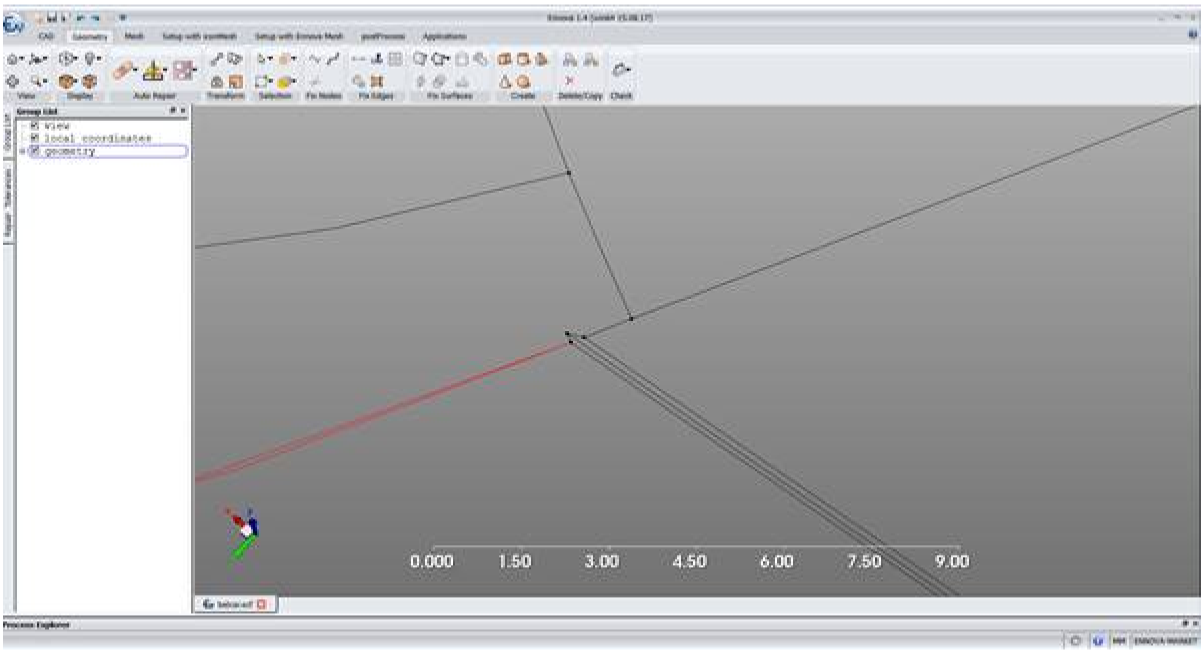
Select the resulting triangle surface and **Remove**.



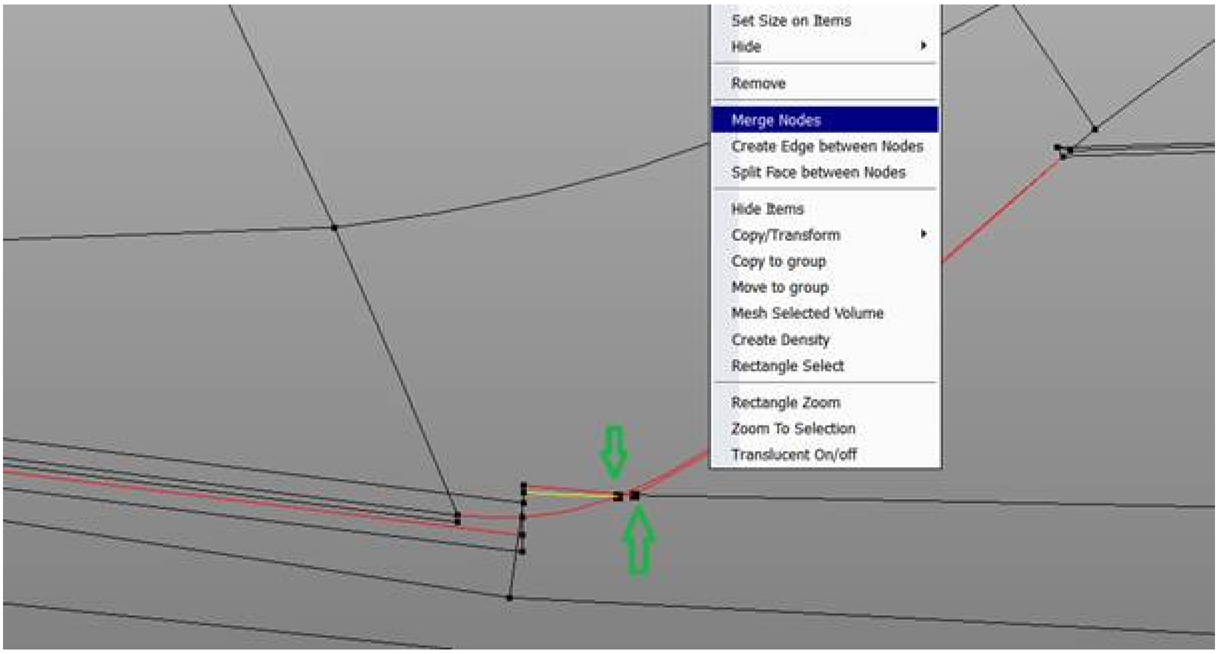
Then select long face edge (RED) and **RMB -> Create Face from Single Edges**.



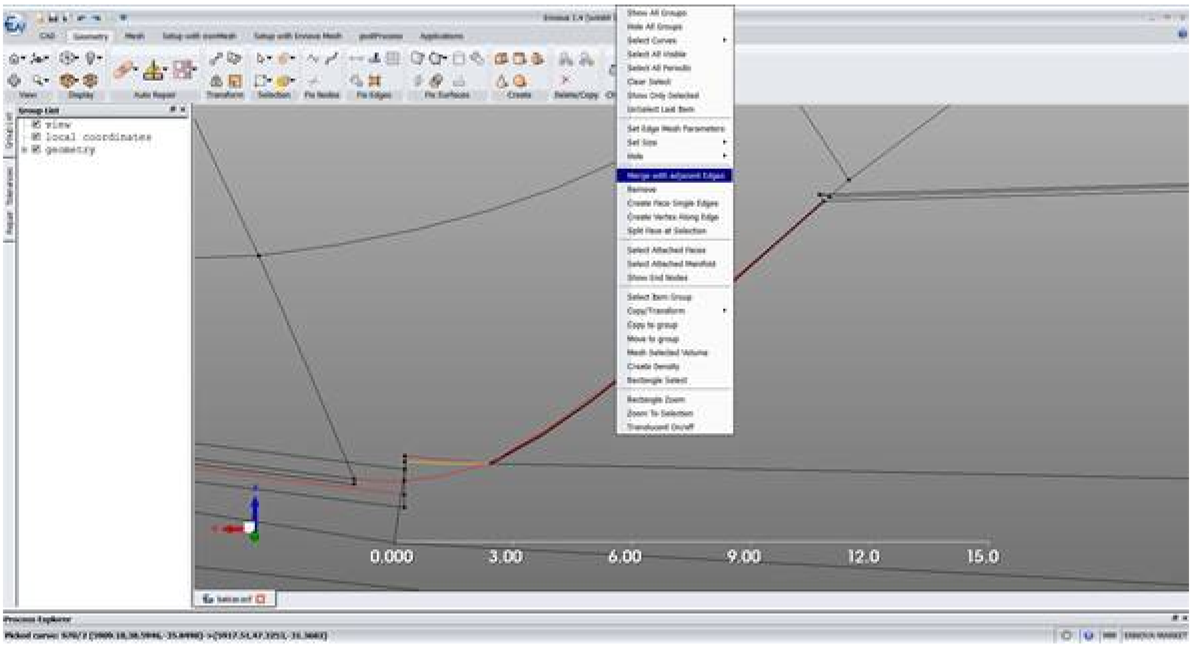
Now the area should be locally watertight (BLACK).



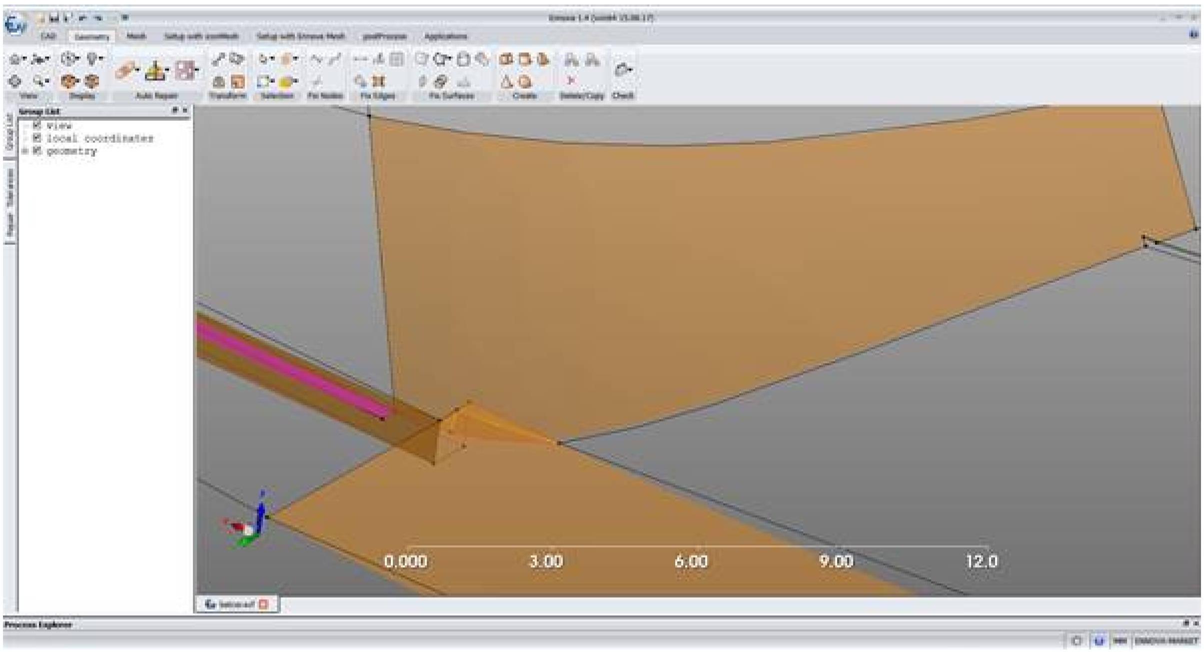
Merge the two nodes at the end of the RED edges as these surfaces need to be forced to connect. Remember as we are working on the topology, even if the CAD surfaces do not connect in real life, in our mesh space we simply declare them to be connected. This is one of the principle advantages of performing CAD repair in Ennova versus a traditional CAD engine.



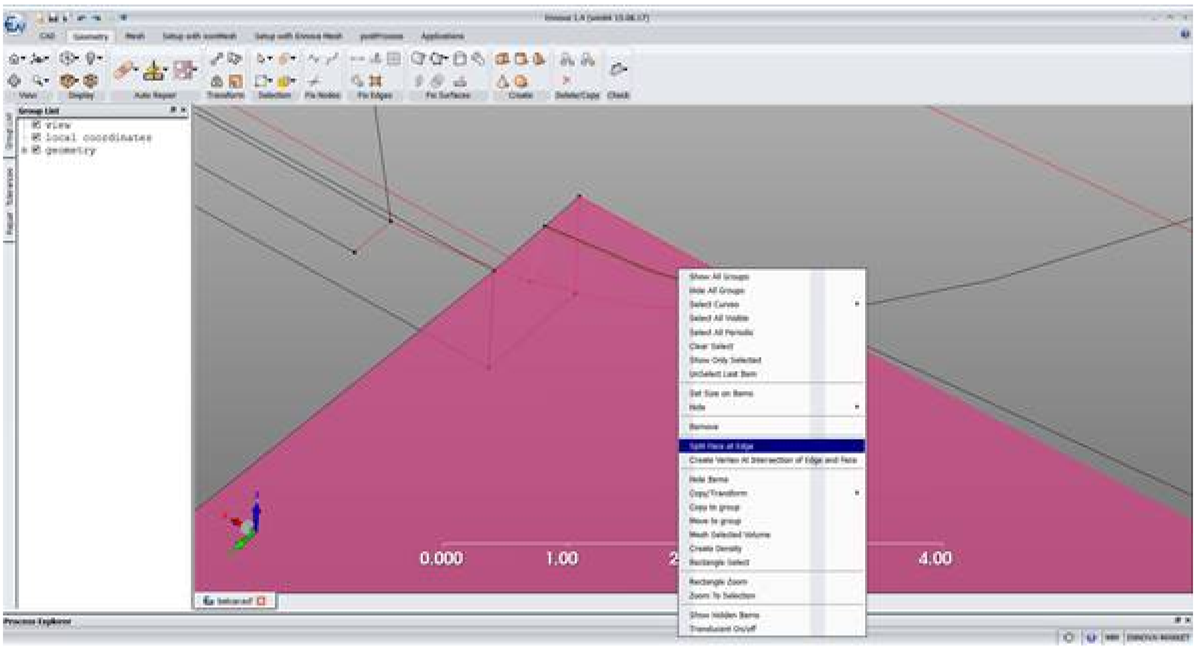
Once the nodes are connected, use **RMB -> Merge with Adjacent Edges** to connect the surfaces.



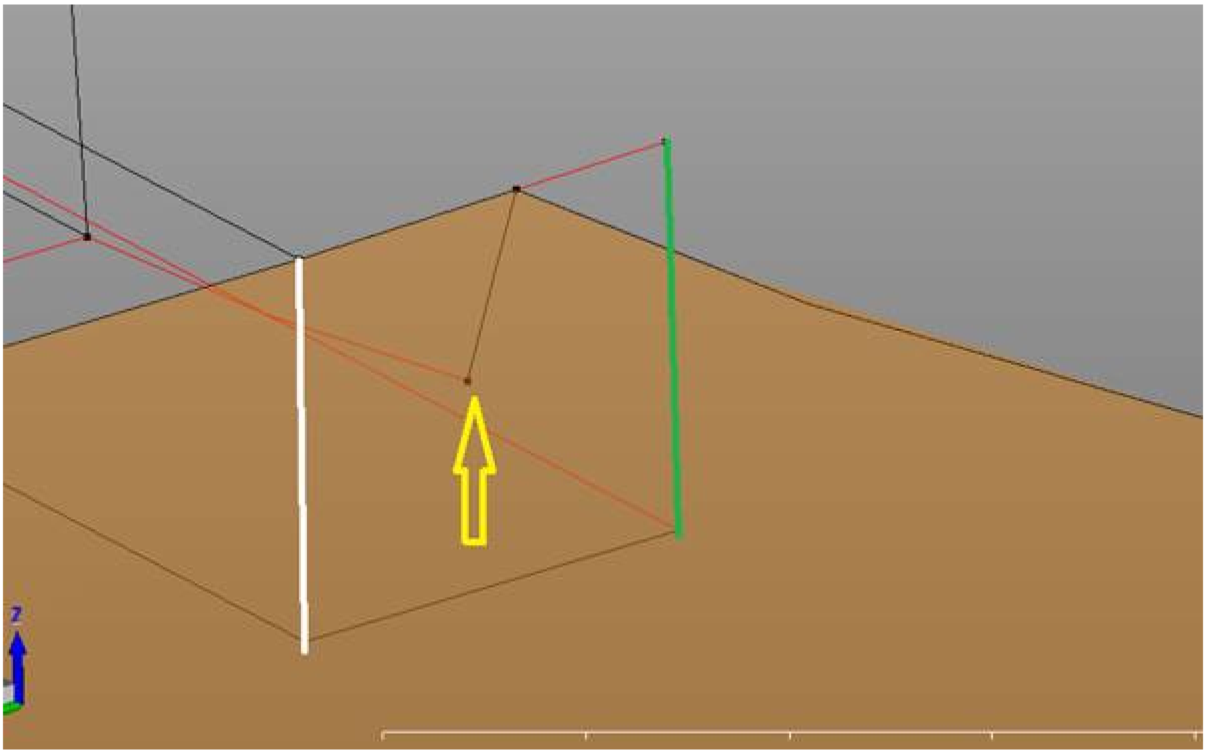
Next we will finalize the front end of the work we did in Remedy 4 and remove any excess trimmed surfaces. Looking at the geometry (still in the region of the last green circle), we see the highlighted surface is not long enough to reach the front of the trench between the cockpit and the body. So we will project nodes, create edges and finally create a face and then perform splits exactly the same as in the rear.



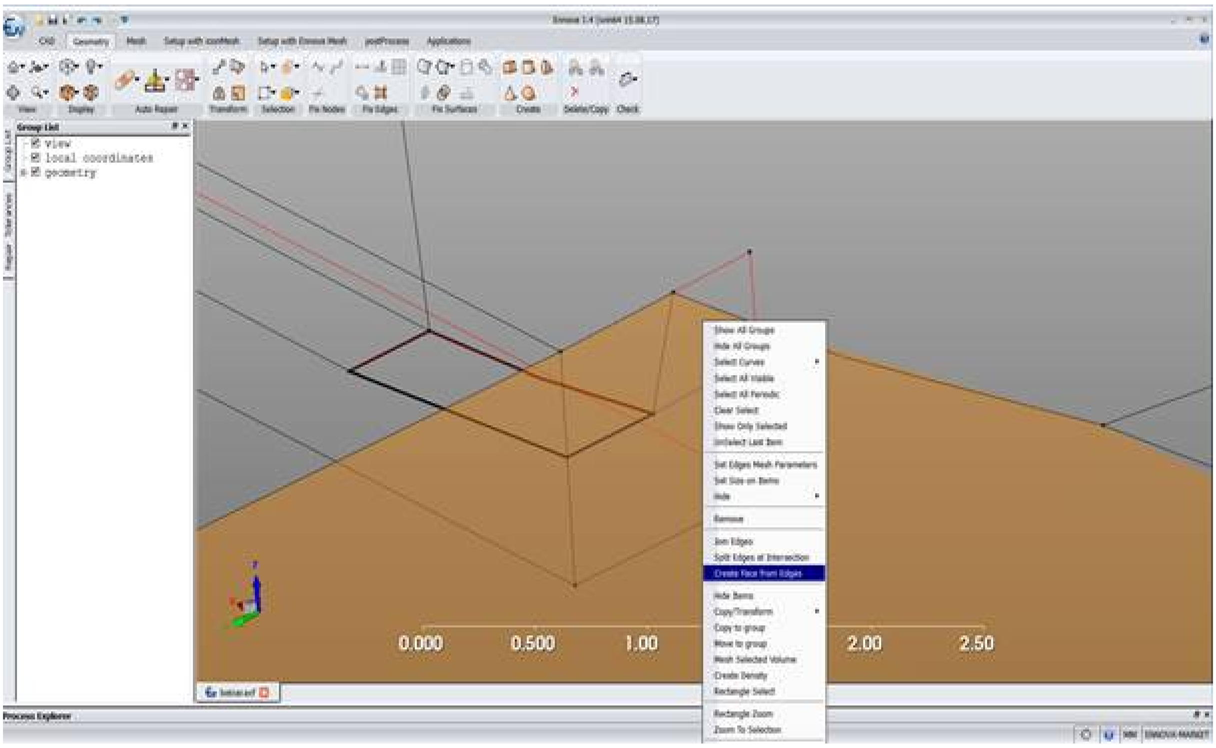
Start by splitting out the corner surface. Remove the two surface “ears”.



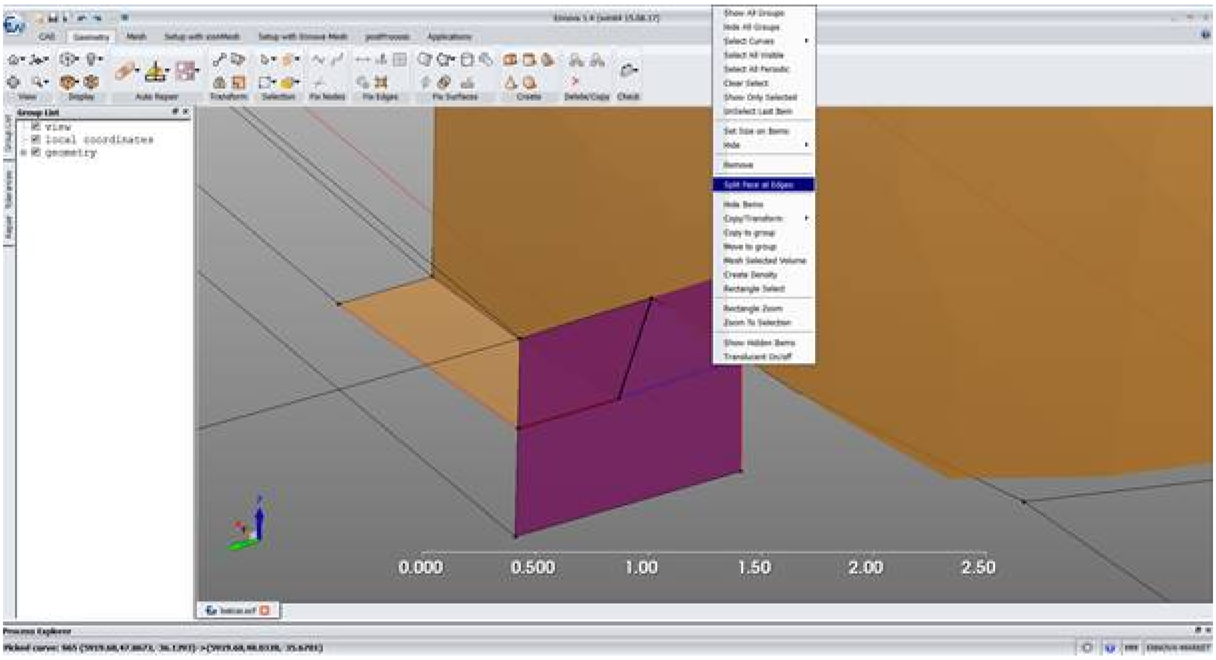
Finally, following the green arrows, project the node to the two edges.



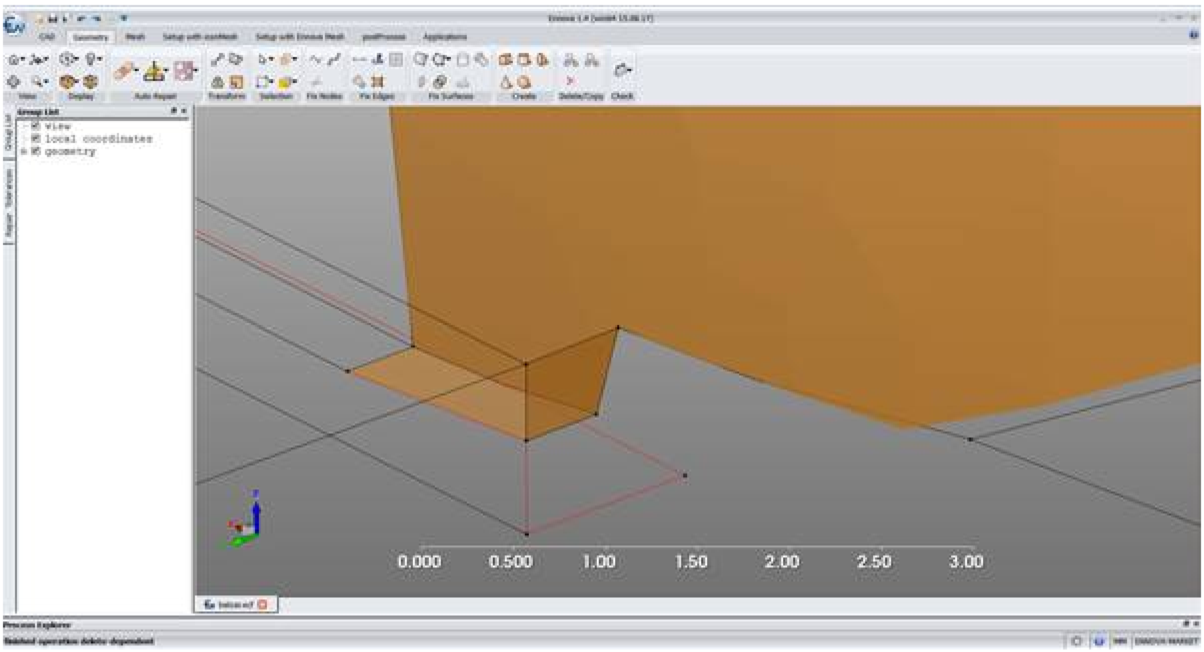
Create edges through the new and old nodes and then create the missing face.



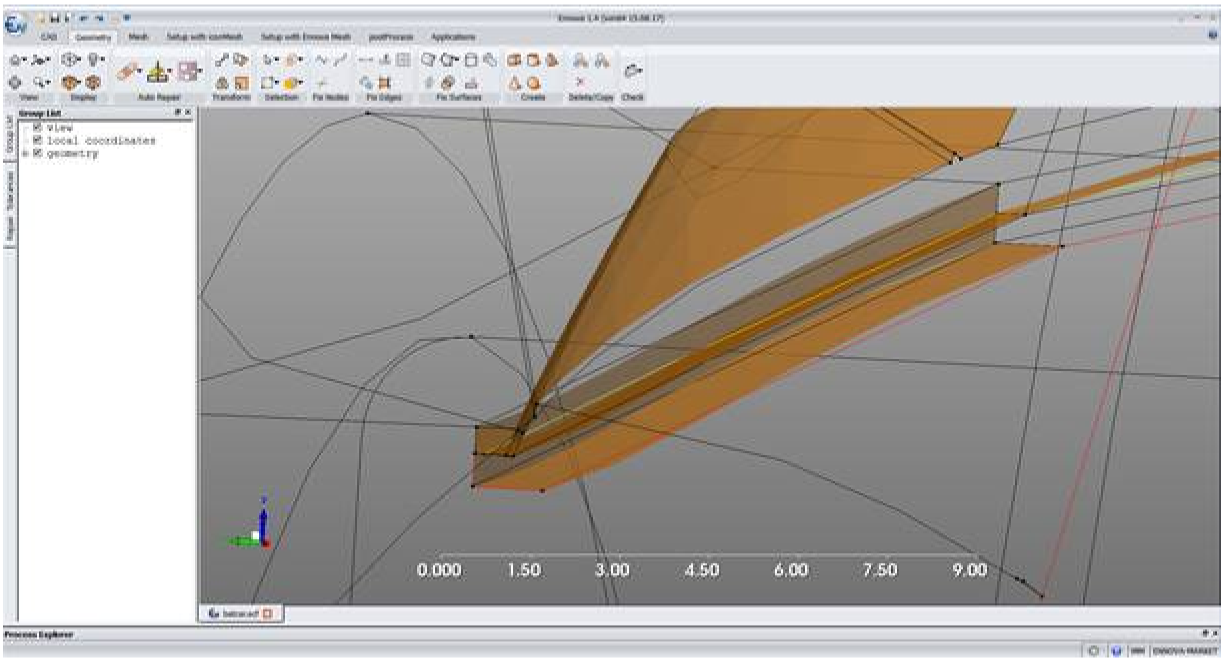
Split the end face corner.



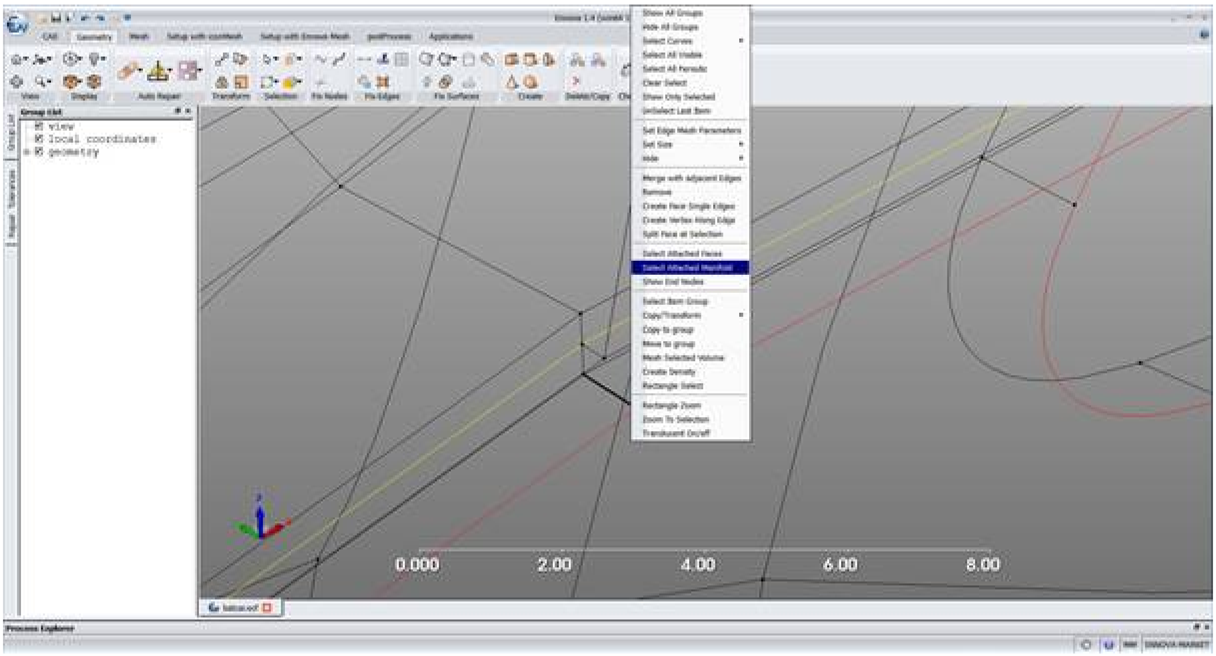
Remove the outside of the unwanted face.



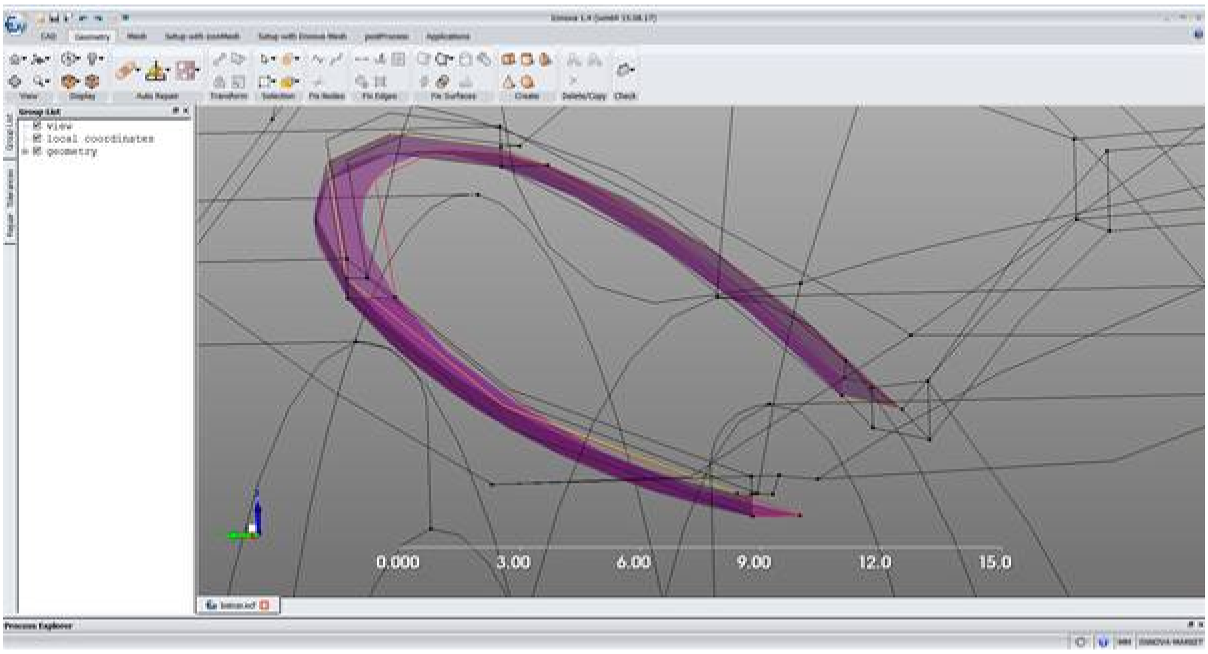
Then split the remaining face along the shelf.



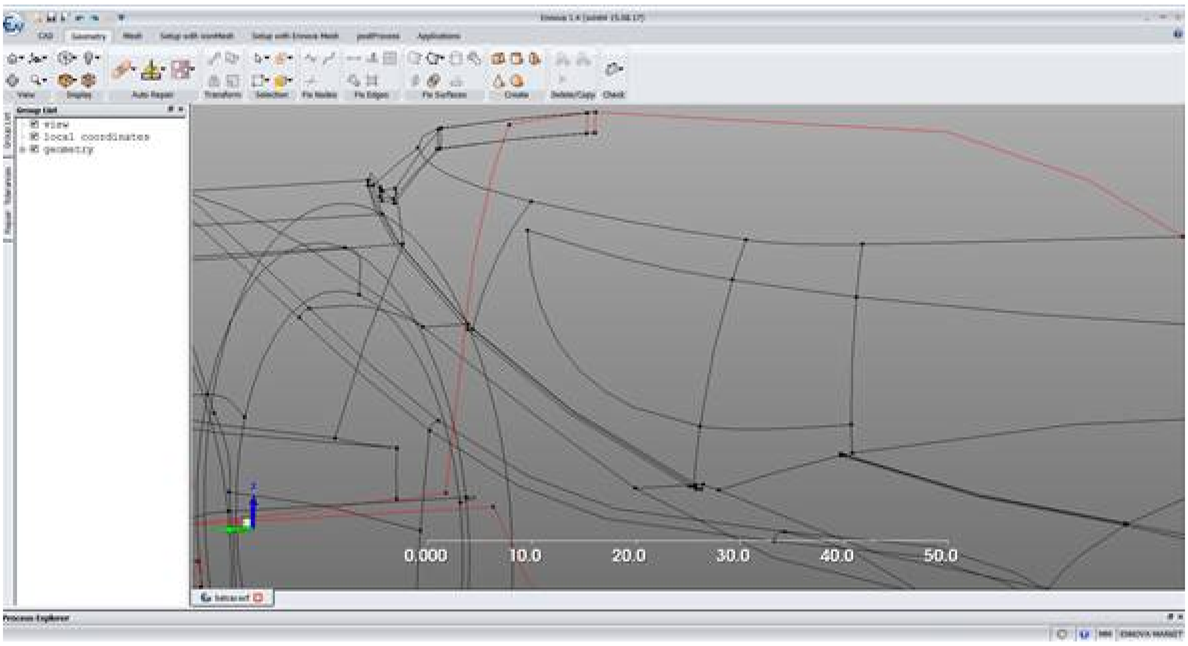
Find a BLACK edge connected to a RED edge on the bottom of the shelf that we split off. Collect all the surfaces that are extra and connect back to the main body via a YELLOW edge by using **RMB -> Select Attached Manifold**.



Make sure you have what is highlighted and then remove this manifold.

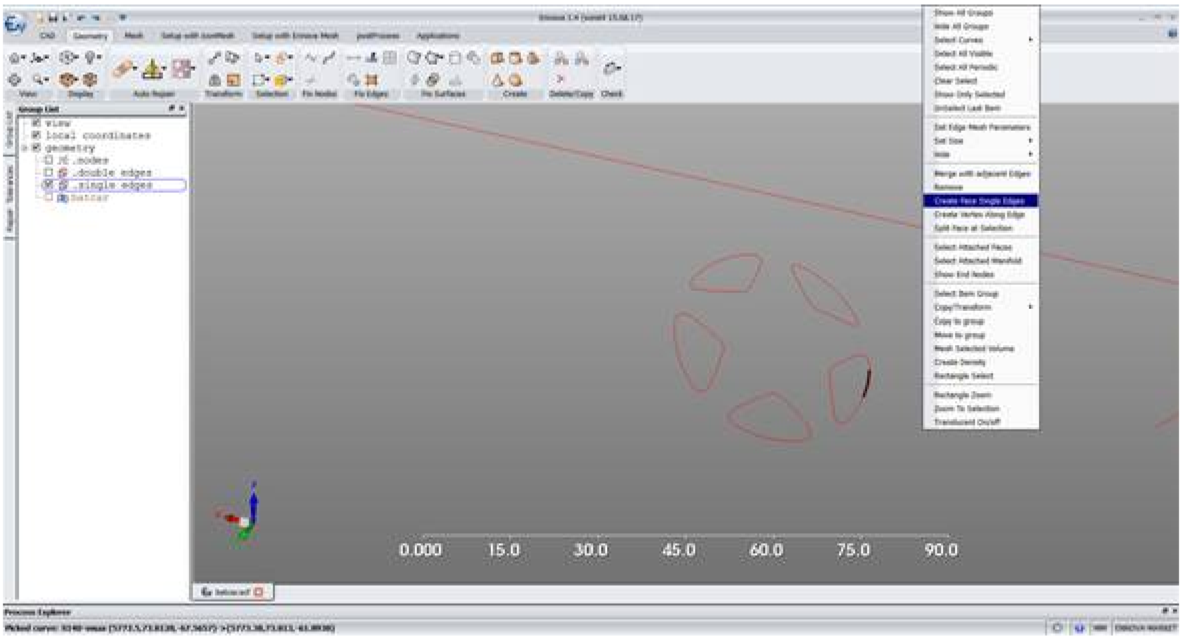


The area should now be BLACK indicating that all is locally watertight.

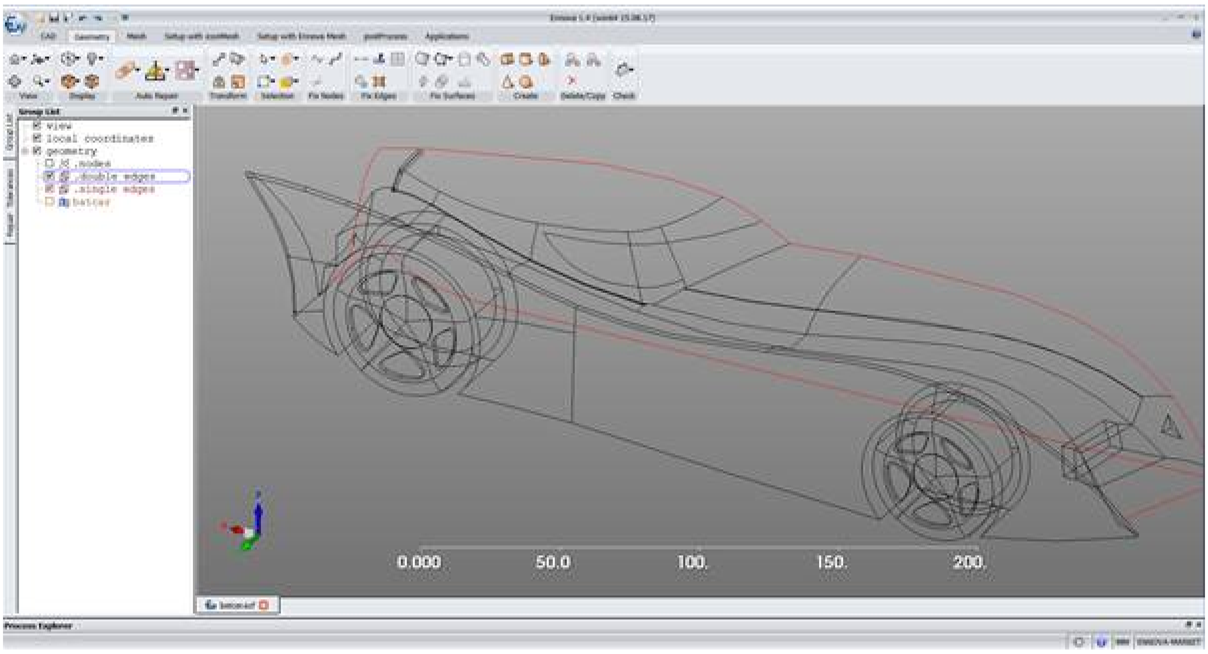


Resolving Initial Results – Remedy 7
Result: Unfilled Loops in the Wheels

Use **RMB -> Create Face Single Edge** to fill all the loops in both wheels. There are 10 loops that need to be filled.

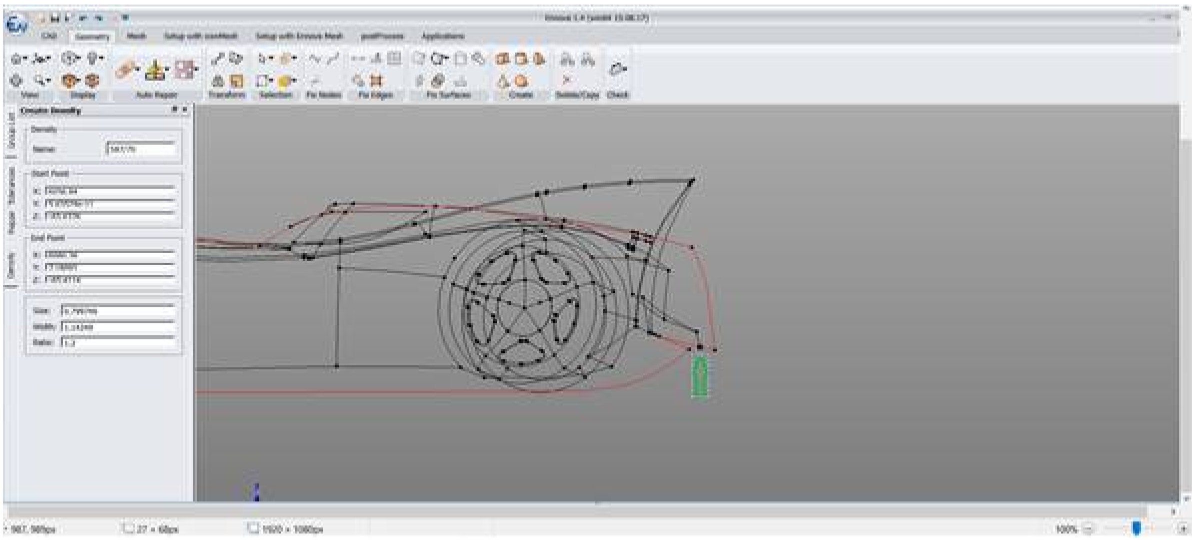


Hint: On the group list pane, try turning off nodes, double and multiple edges. That way, only RED lines are visible.

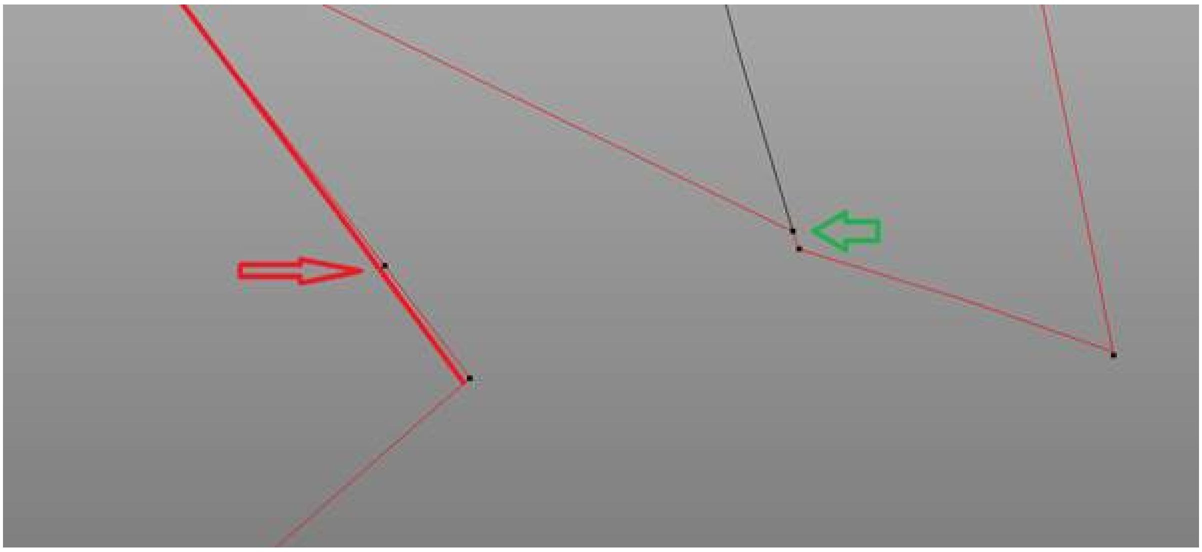


Resolving Initial Results – Remedy 8
Result: Gaps Between Floor and Car on Symmetry Plane

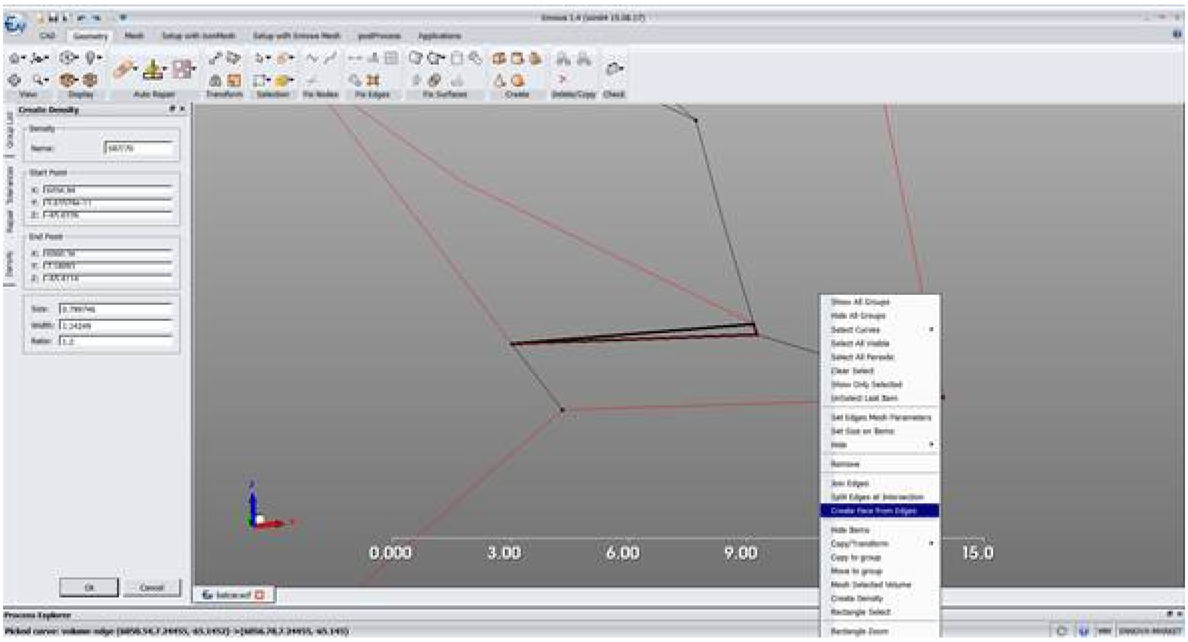
Zoom in on the rear of the car and use **RMB -> Create Vertex on Edge** at the corner of the edge that does not have a vertex.



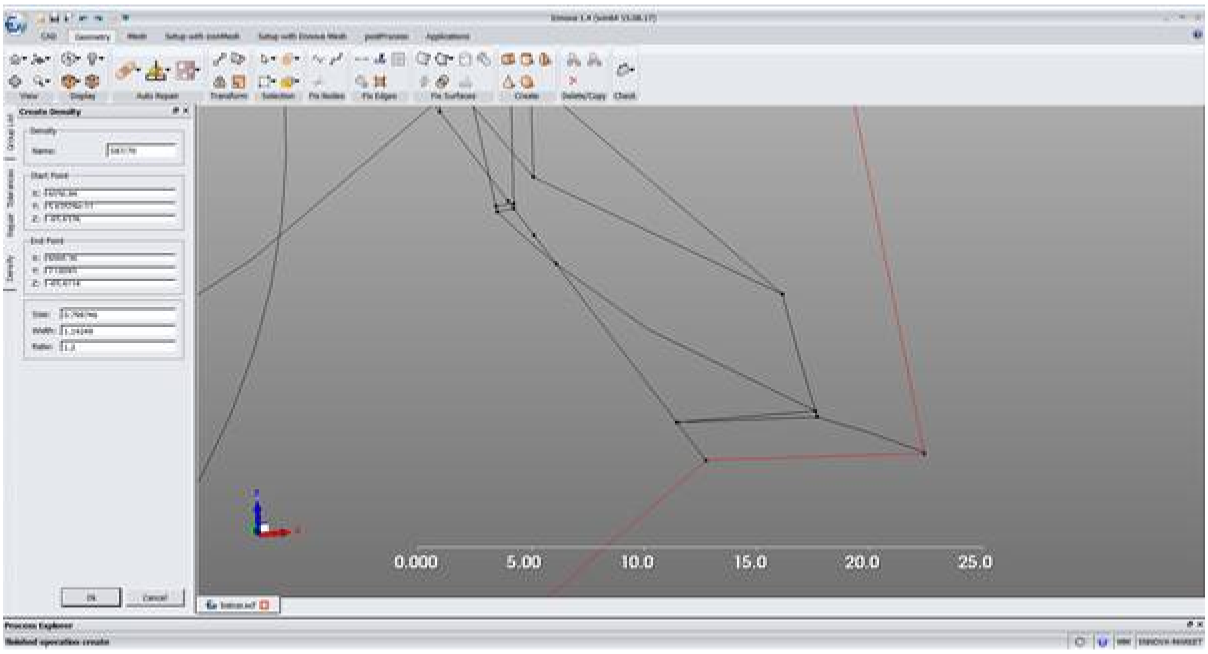
Next, use **RMB -> Project Vertex to Edge** by selecting the Green arrow vertex and RED arrow edge to make the new vertex. Then create the missing edges and faces to fill the gaps.



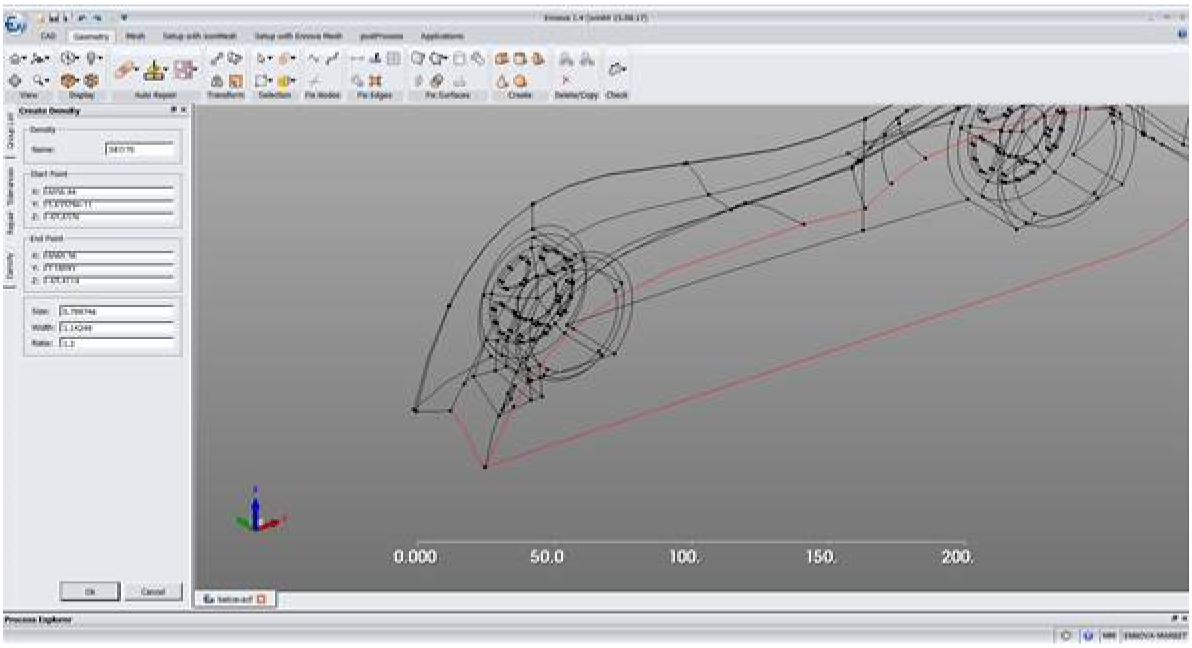
Fill in the three faces including the triangle step face.



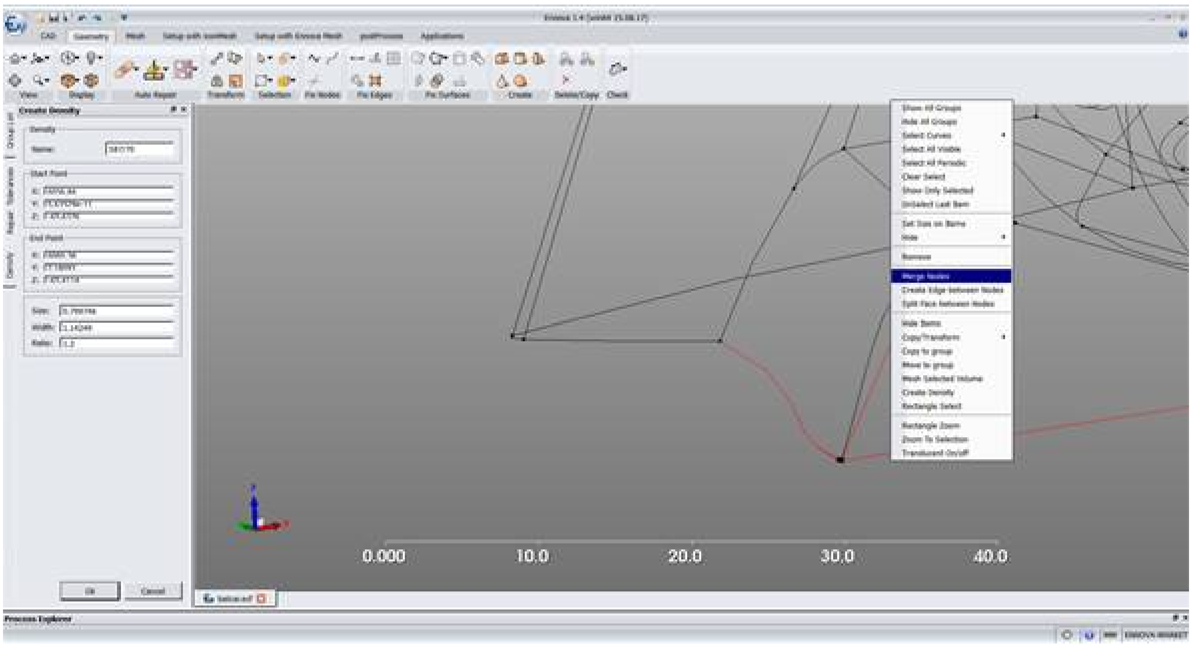
Now the region should be locally waterproof.



On the front end of the car the floor/body intersection is not valid as indicated by an extra RED edge. Start by merging the end nodes.

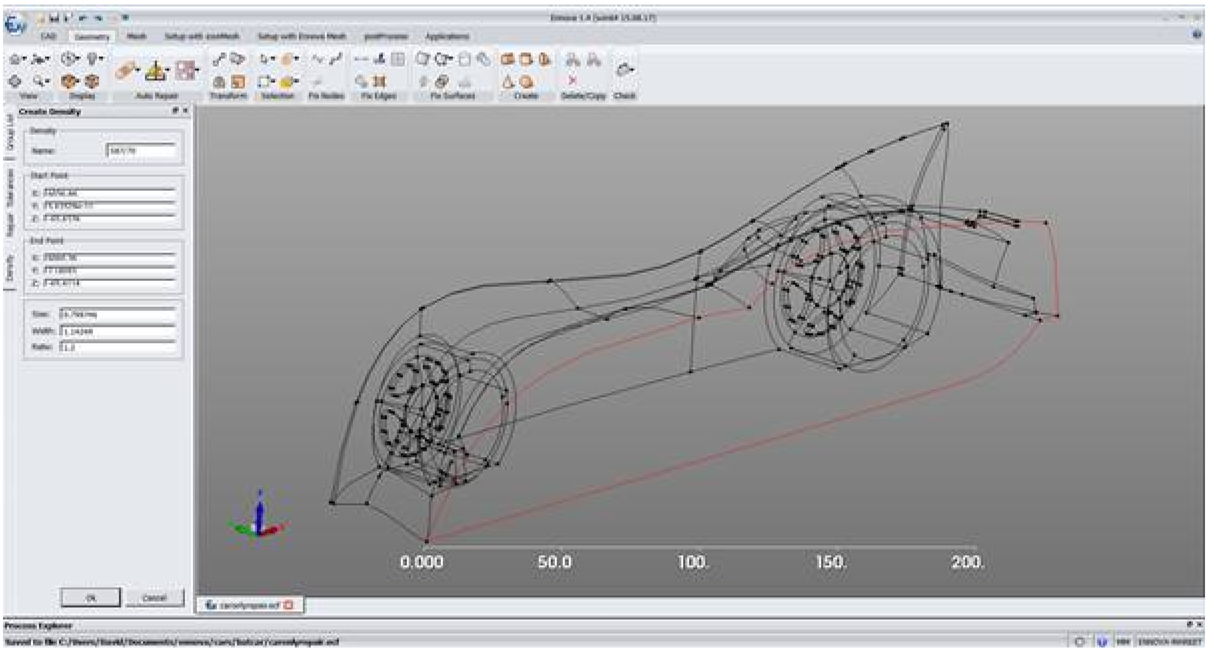


Then, one of the edges will share both end nodes. Now use **RMB -> Merge Adjacent Edges**.



BATCAR CAD Repair Complete

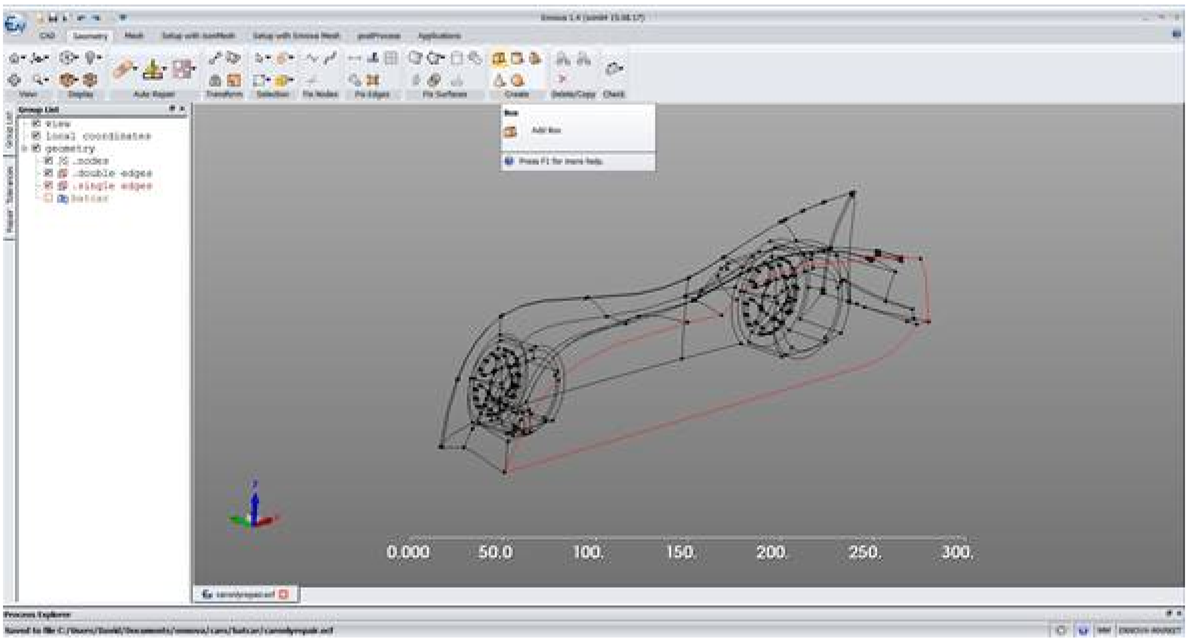
That concludes the repair of the car. This stage of the tutorial model is saved as caronlyrepair.ecf. Notice that the car is all BLACK except the single edges that will connect to the CFD domain.



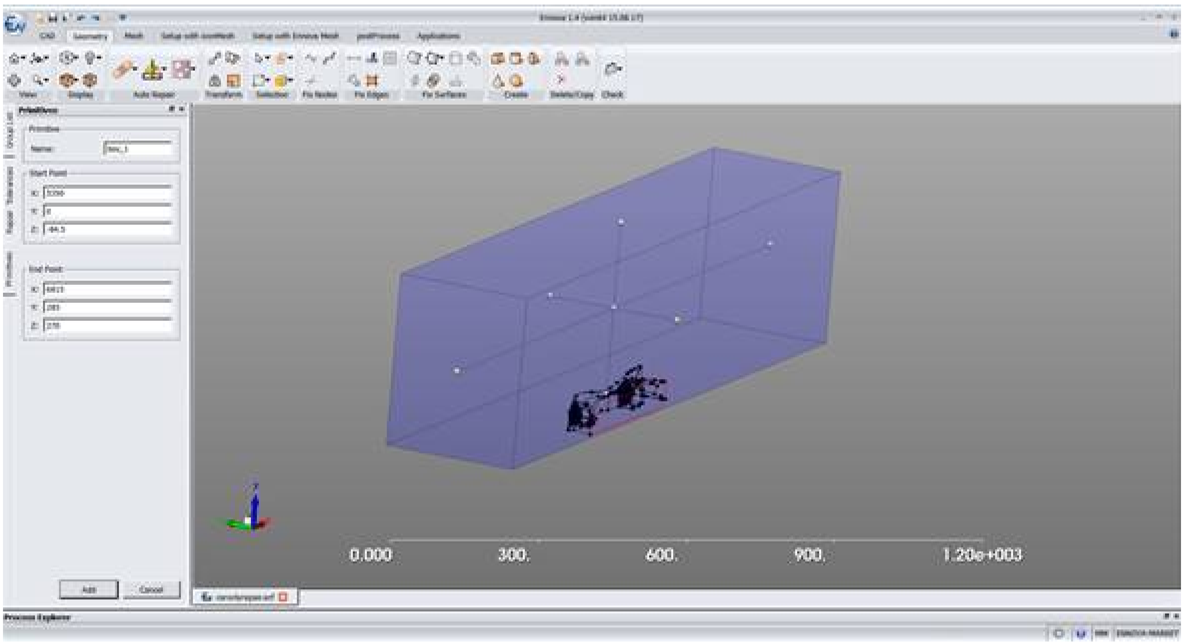
CFD Domain Set Up

Use the Box icon to add a box. Set box to:

- Start Point (5350, 0., -84.5)
- End Point (6815, 285, 270)

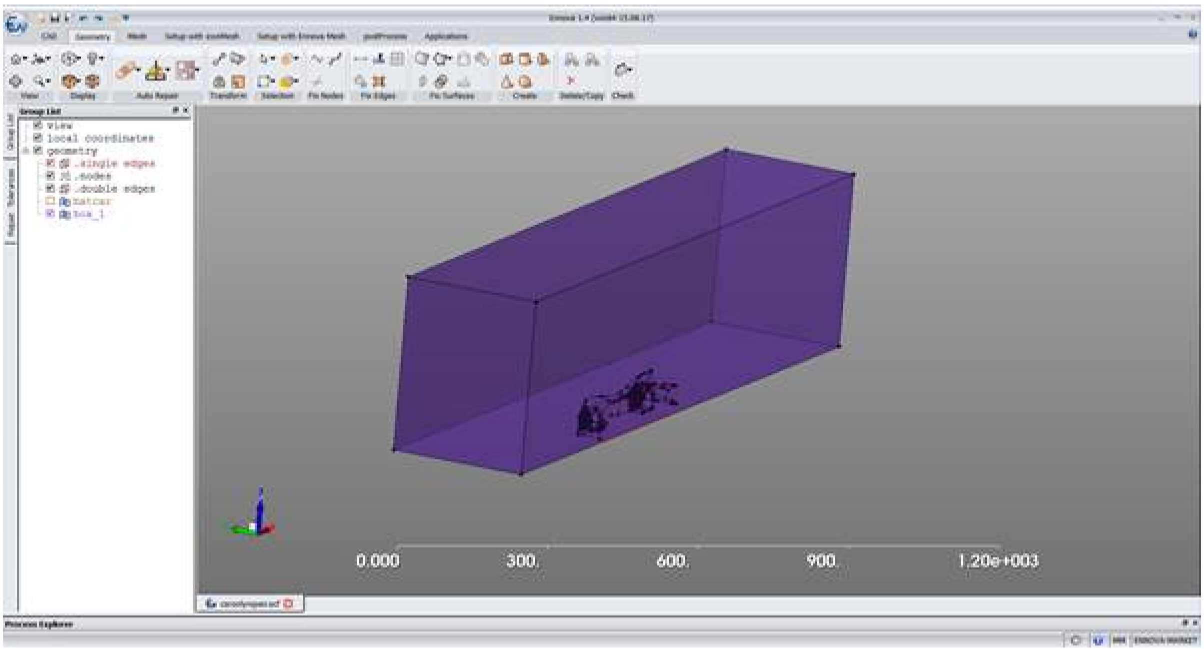


Don't forget to press "add" to finalize the box geometry in the model.

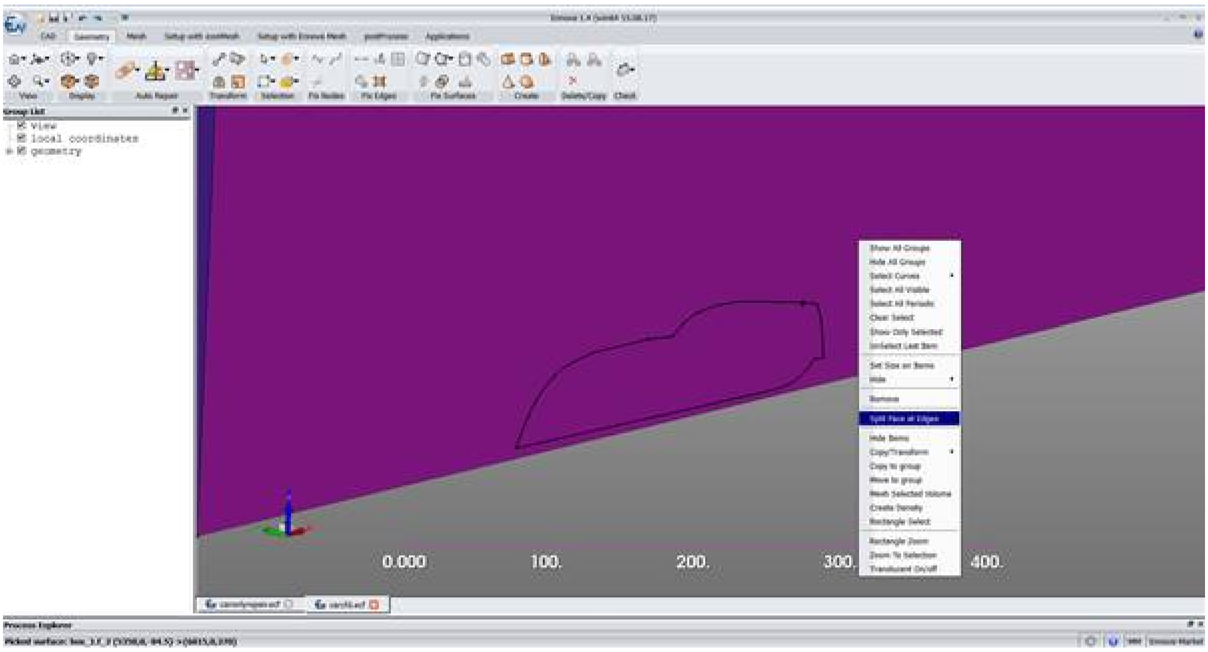


Geometry Repair

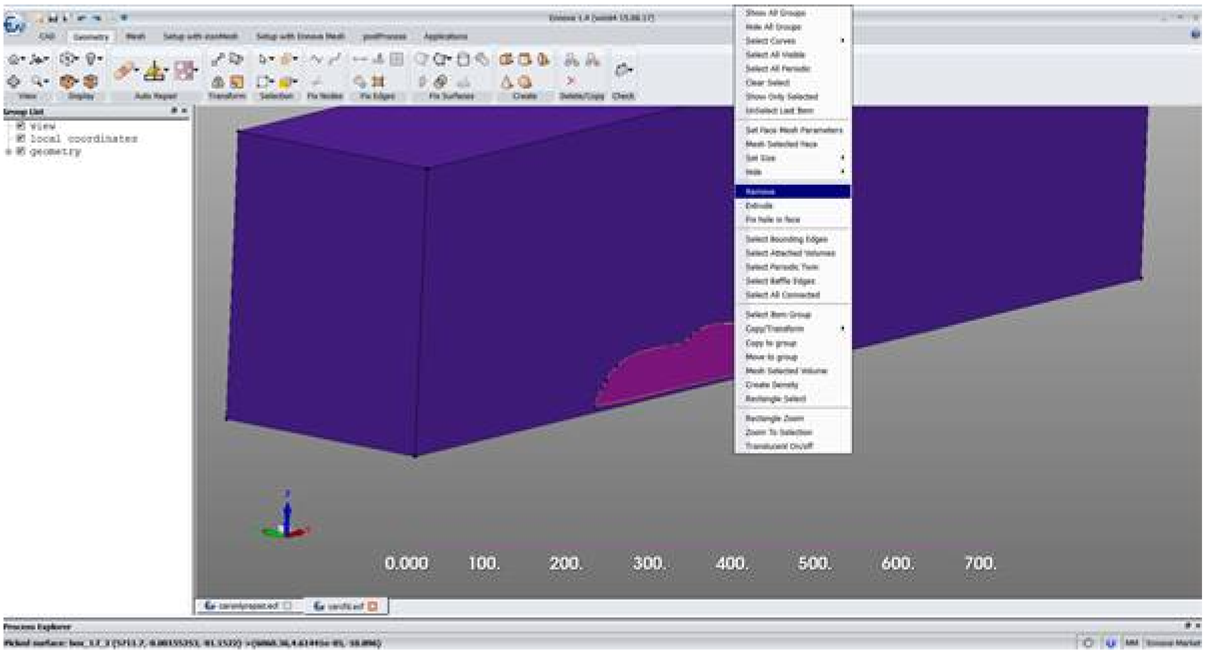
Select the edges of the model that are RED and lie on the symmetry plane and also select the symmetry plane itself. Use **RMB -> Split Face at Edges** to connect the model to the symmetry plane. The edges will turn YELLOW at the connection.



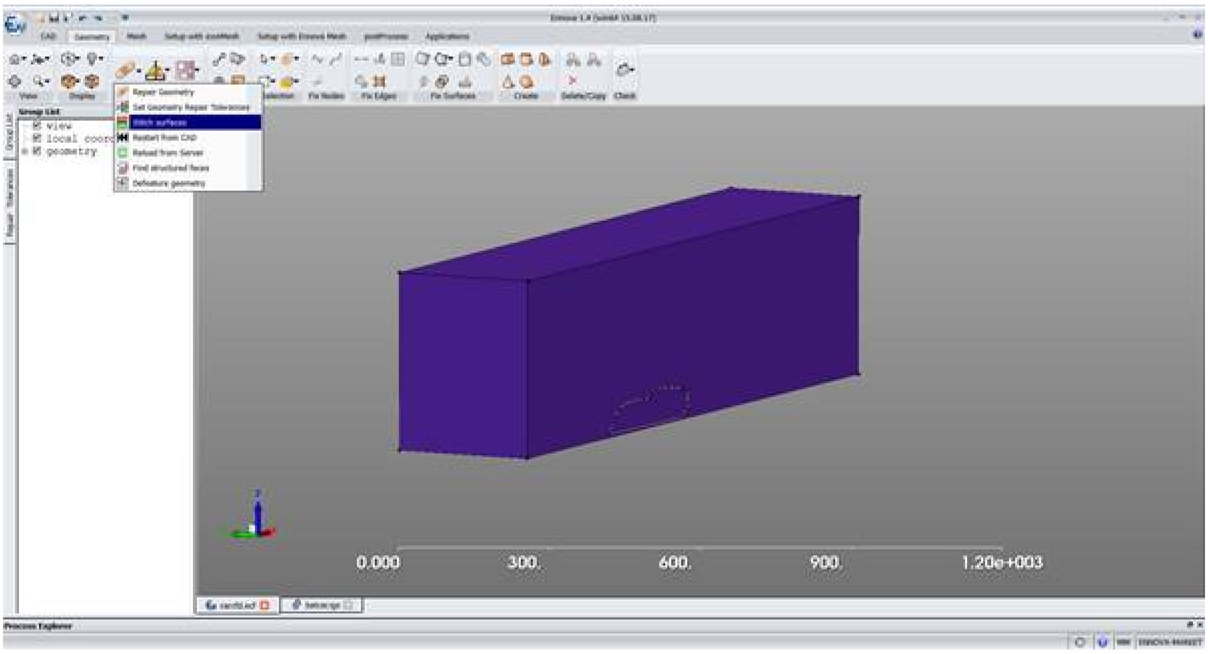
Then select the new middle face. Remove it with **RMB -> Remove.**



Finally, the wheels need to be trimmed to the floor.



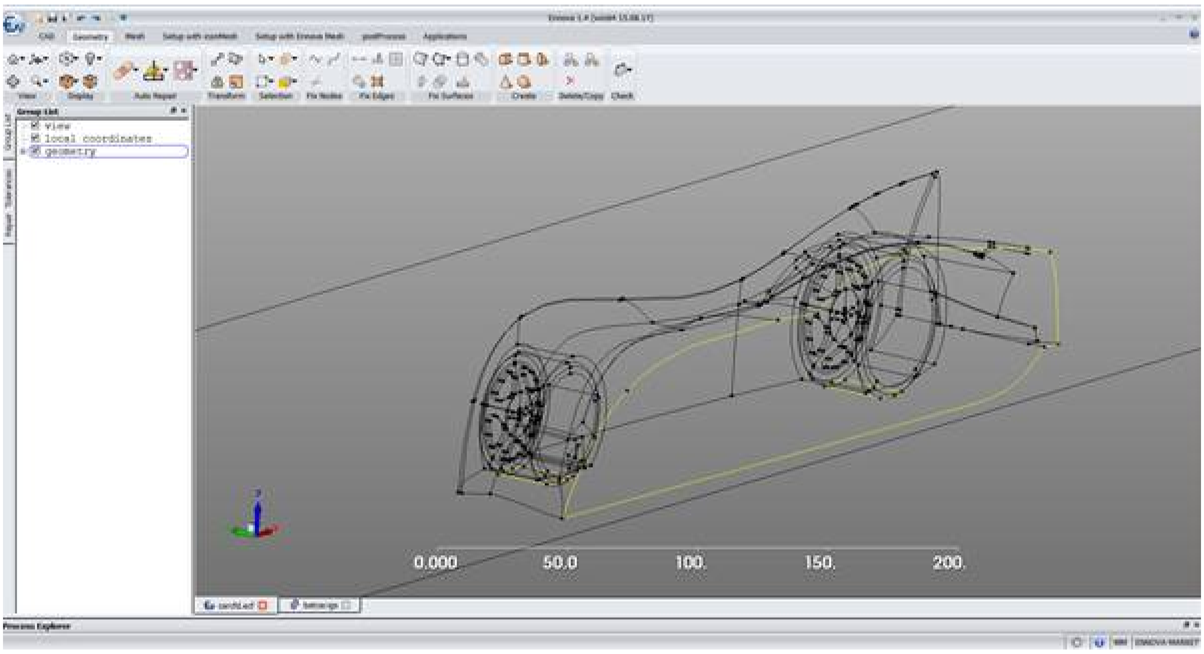
After running the Stitch Surfaces command, the intersection of the wheels and floor will be trimmed.



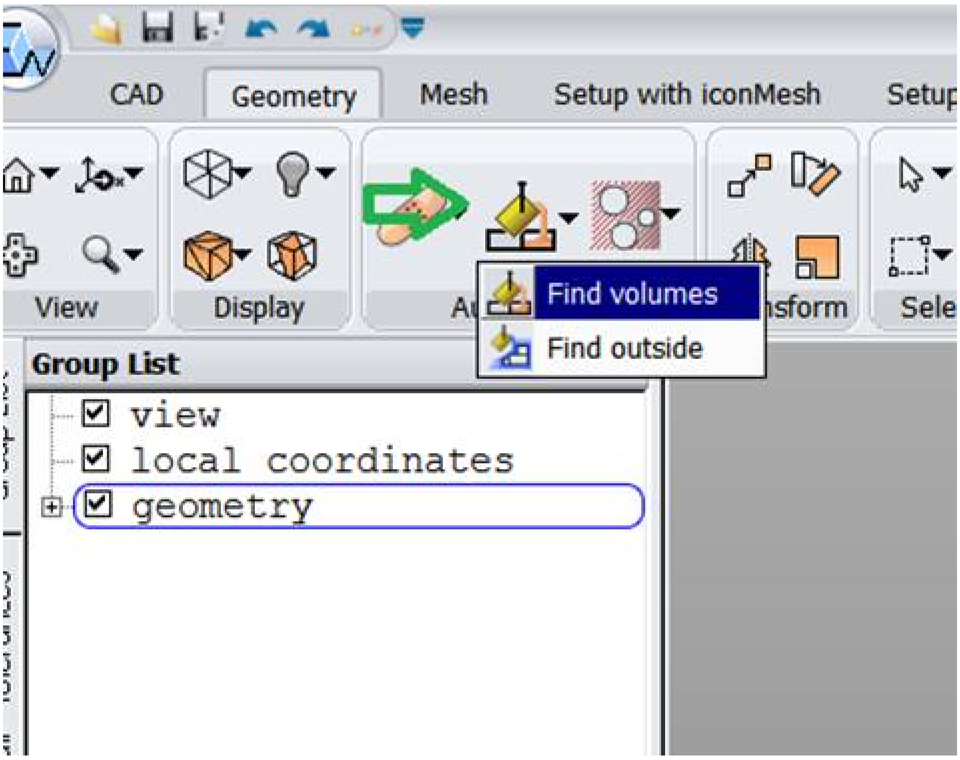
Now the geometry repair is finished.

CFD Volumes

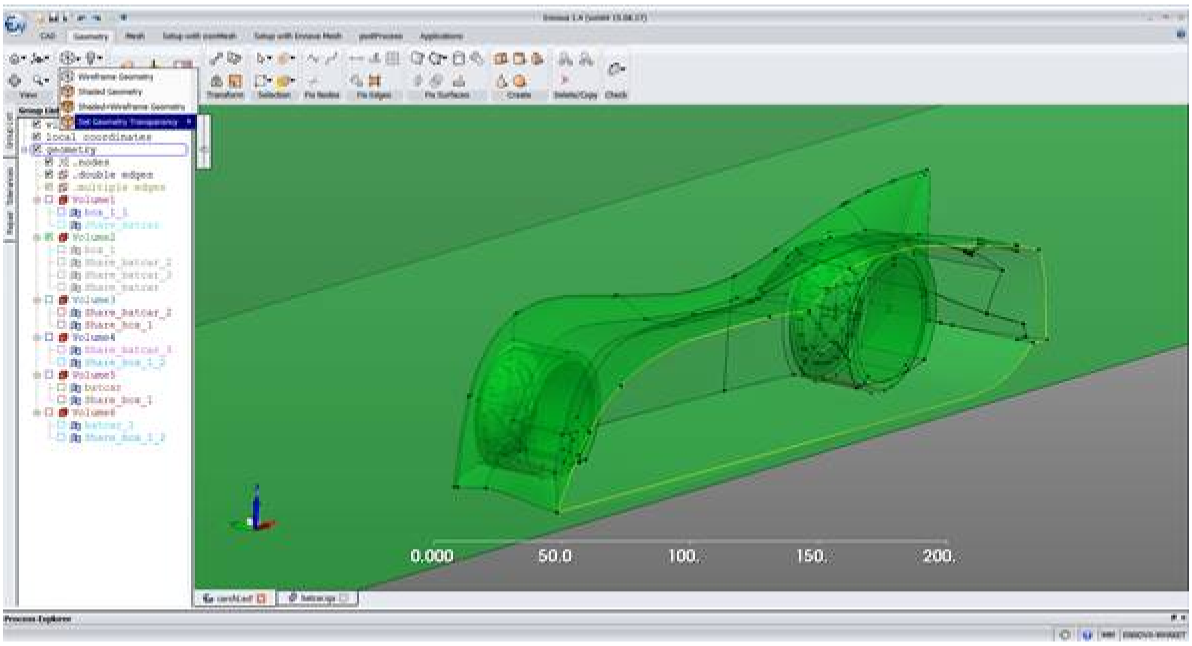
You will now see mainly BLACK edges and three YELLOW loops, one YELLOW loop for each wheel and one for the symmetry plane intersection with the car.

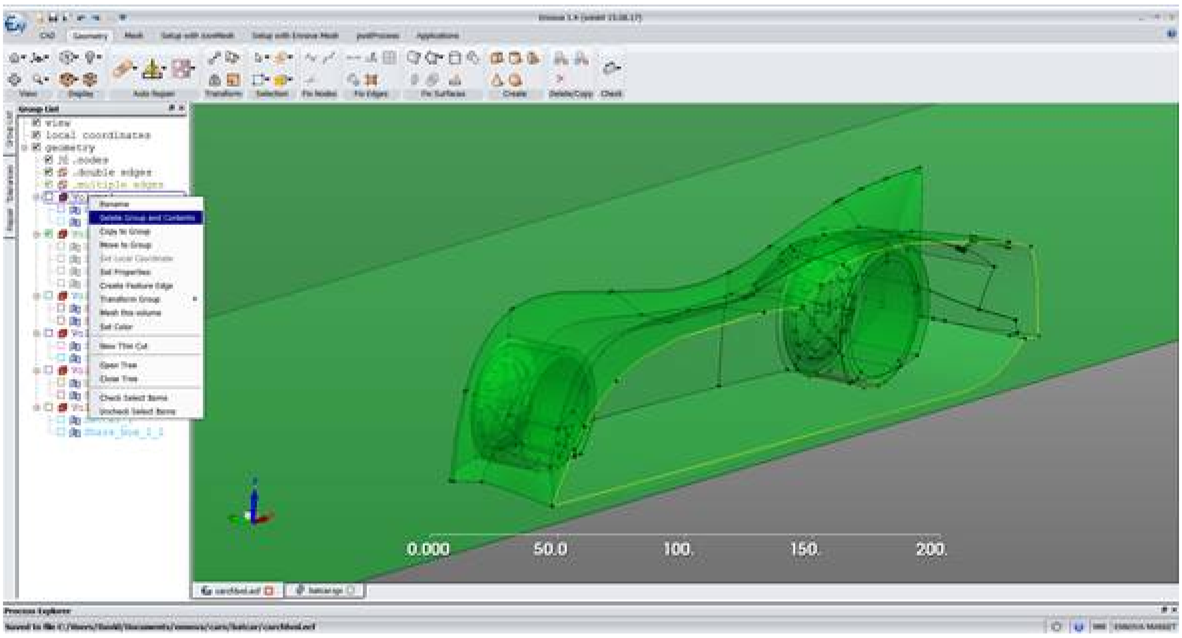


The next step is to extract the CFD volume. Use the **Volume** command to compute the volumes.



RMB on the volume domains is not necessary.
Choose **RMB -> Delete group and contents** for volume 1, 3, 4, 5 and 6.





Ennova finds 6 volumes:

- Two wheel parts inside the box
- Two wheel parts outside the box
- The interior of the car
- The exterior of all the geometry (car + wheels) which is interior to the box

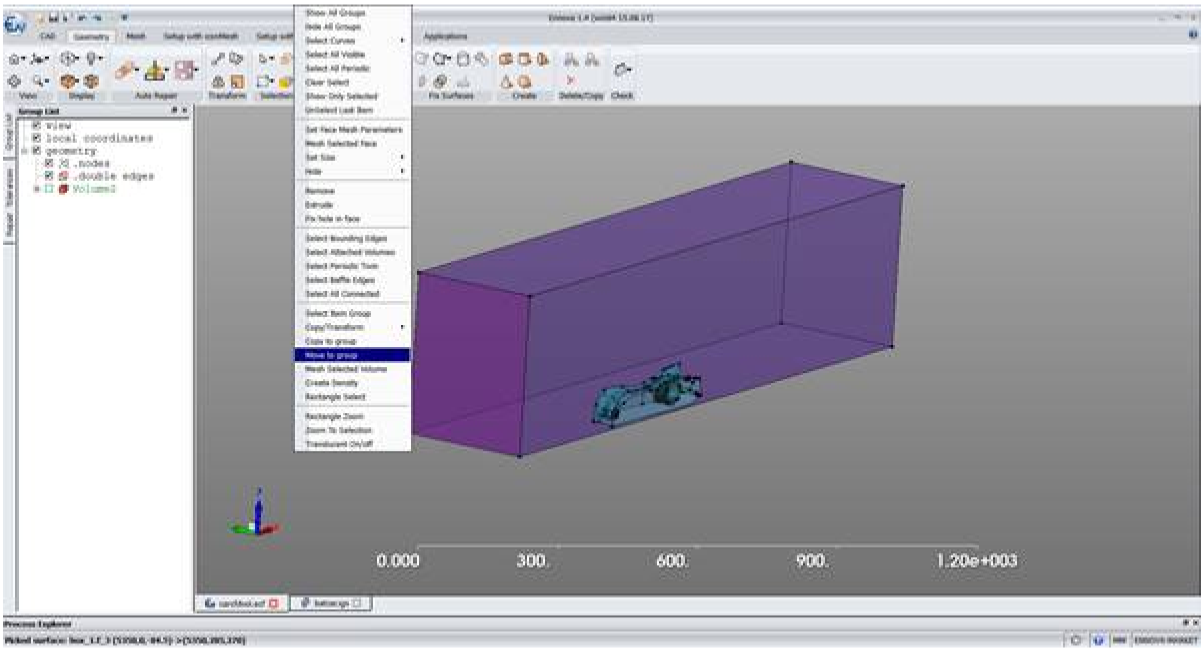
Volume 2 is the CFD domain we want. We will remove the other domains for this analysis.

Boundary Conditions

It is convenient to setup the boundary conditions next.

- Inlet
- Outlet
- Tunnel Walls
- Floor

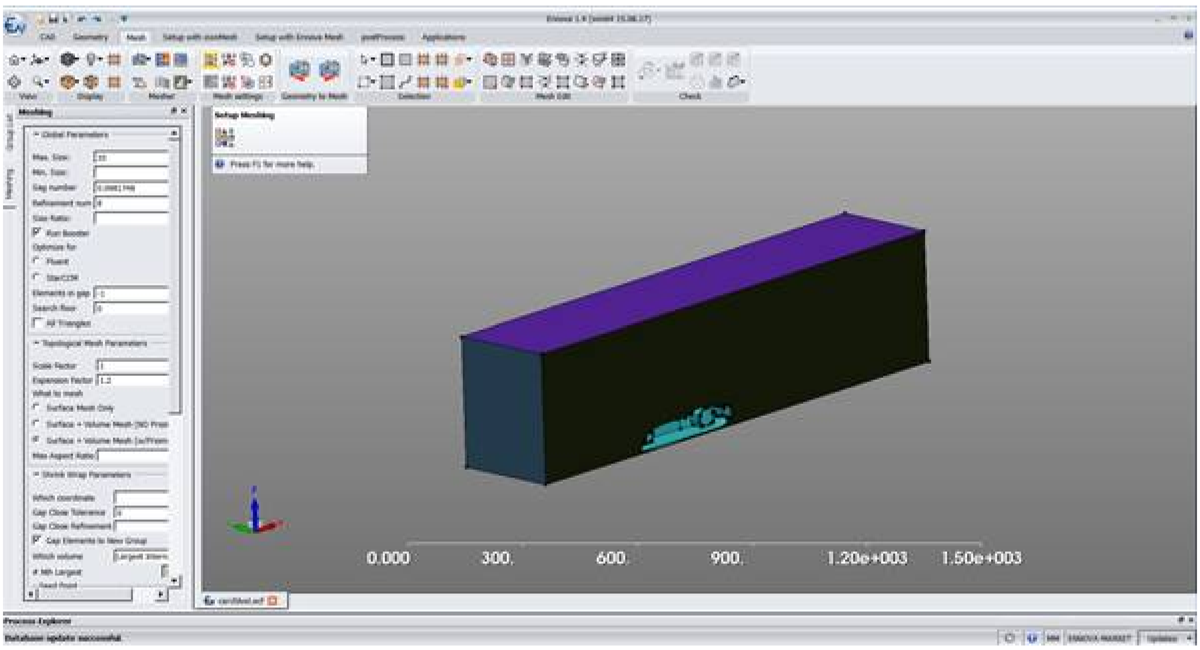
Select each surface in turn and use **RMB -> Move to Group** to rename each boundary.



Mesh Control

Ennova uses a parent child hierarchy as the control for the mesh. If nothing is set for the child, then the child's parent is used for control. If the parent is not set, then the parent's parent is used, etc. The top default levels are the global parameters.

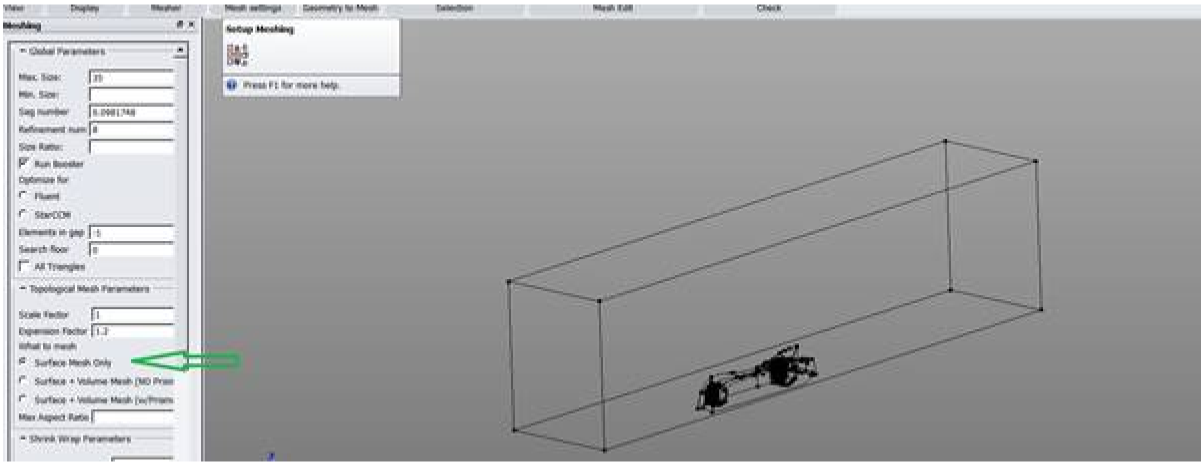
We will start by setting the global maximum mesh to size 35. Ennova uses many clever algorithms to set mesh sizing and usually choosing a reasonable maximum size is sufficient to get a good starting mesh.



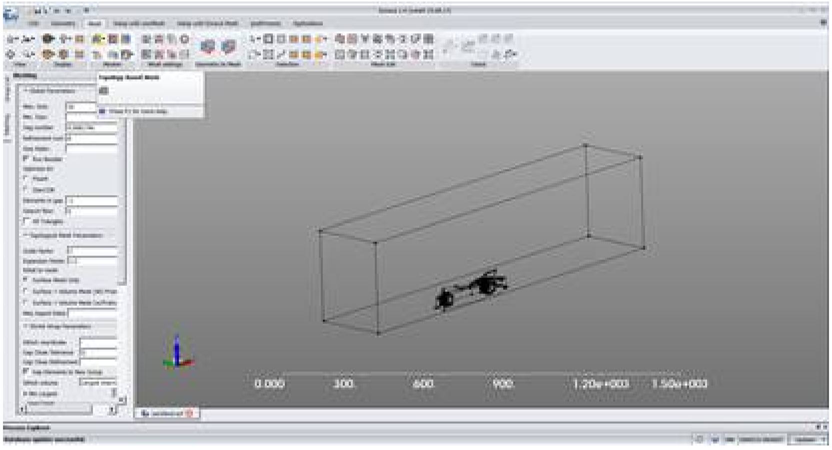
Surface Mesh

First we can test a mesh to have a look at our mesh sizing and to test its quality. Obtaining a smooth error free surface mesh is key to obtaining a good CFD volume mesh.

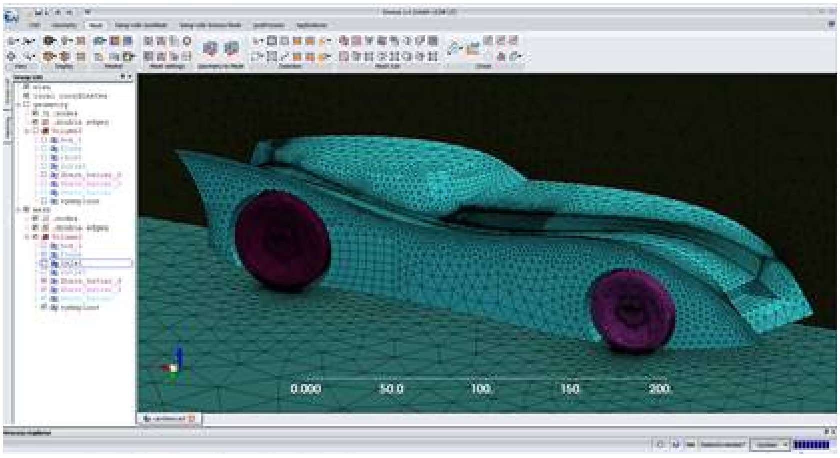
Set the **Surface Mesh Only** radio button.



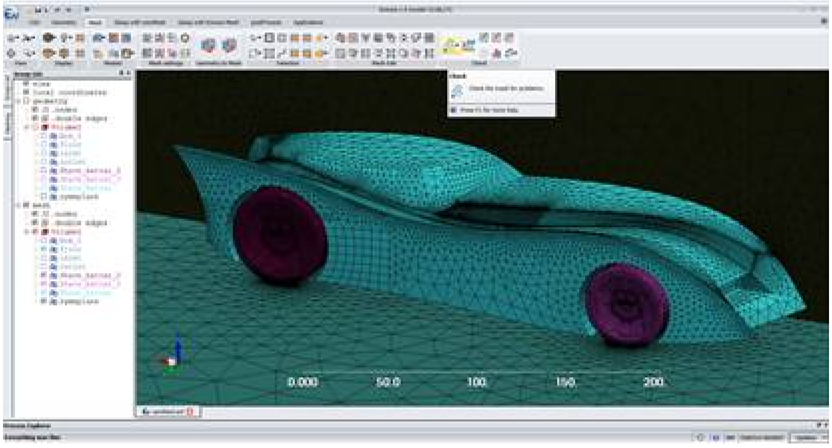
Generate the surface mesh by clicking the **Topology Mesh** icon.



Use the stethoscope to check the surface mesh.

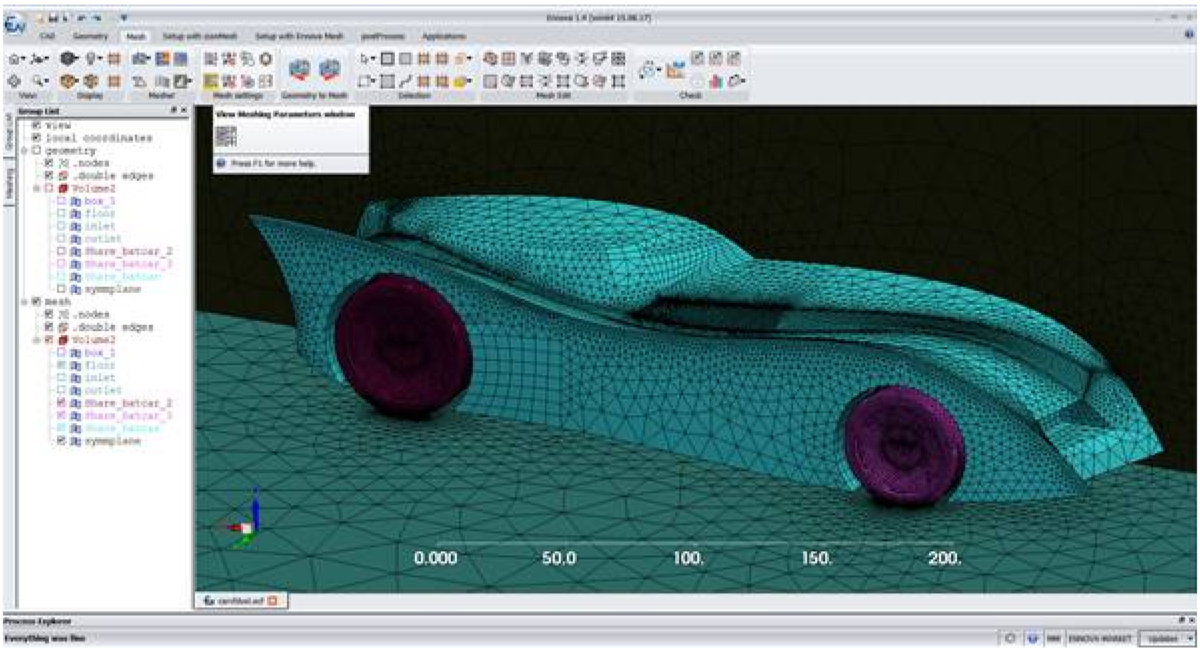


It will report that the mesh passes all tests.

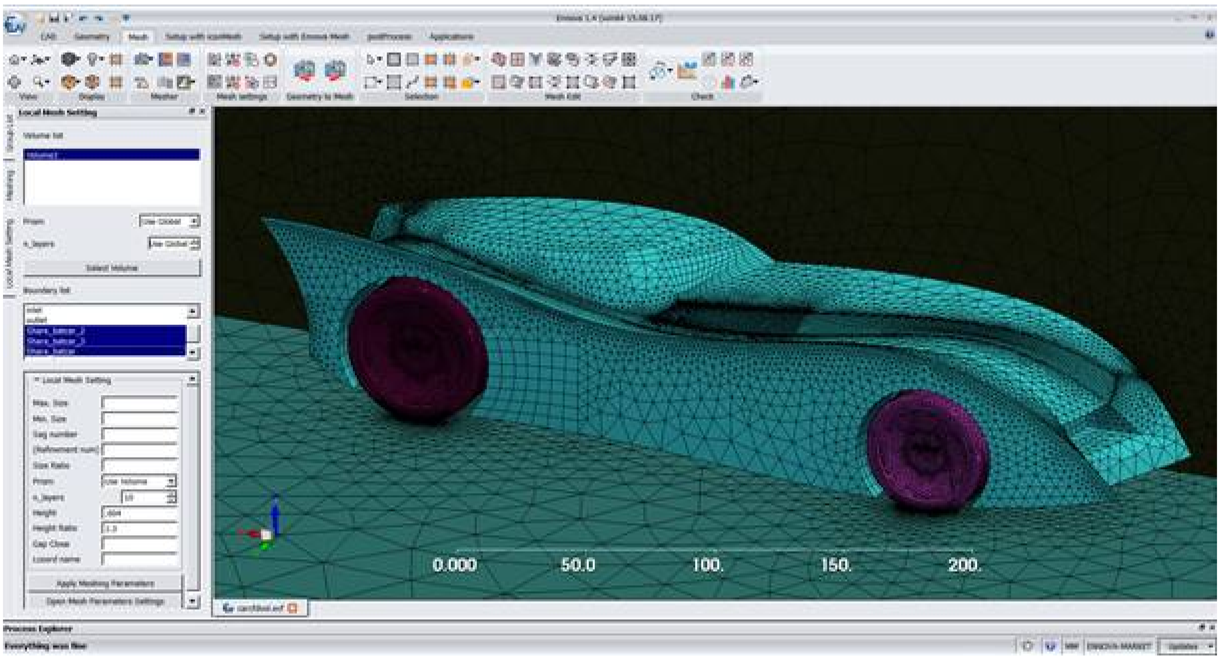


Set Up Boundary Layer Prisms

In Ennova there are several ways to input and control the prism layers. Today we will use the dropdown box to control from which surfaces we will extrude prism layers. We will then check the input using the spreadsheet.



Use the **View Meshing Parameter** icon to get to the prism control.



Prism Control

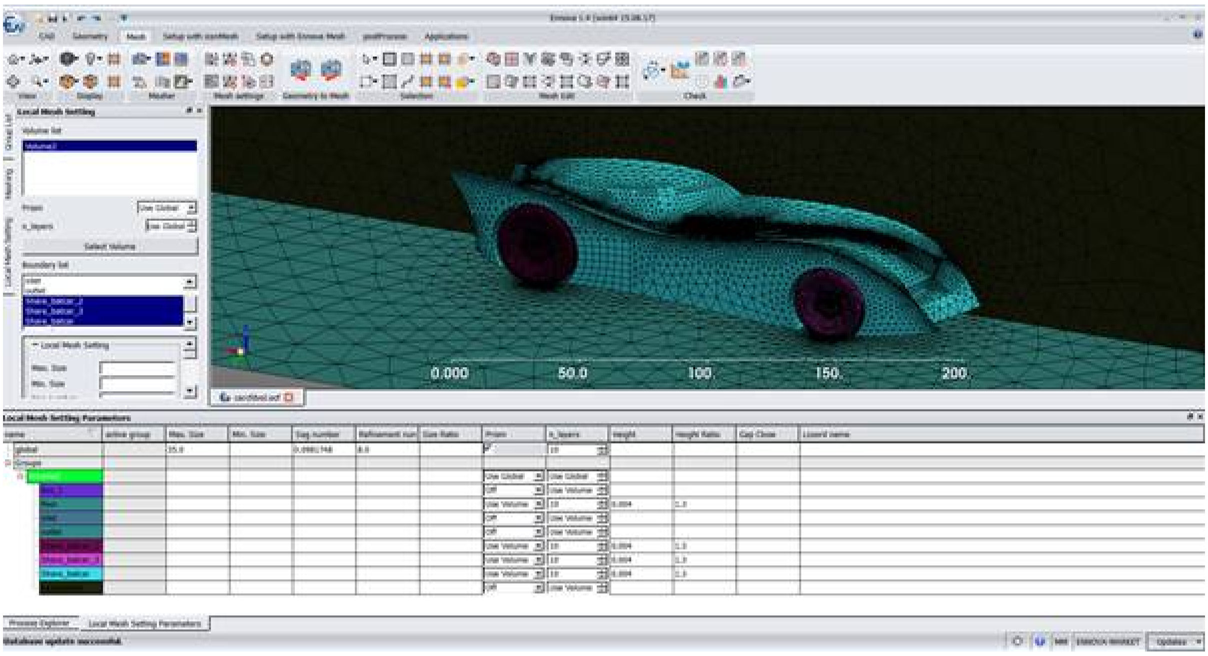
In the Control, highlight the volume and the groups that need prisms.

- Floor
- BATCAR
- BATCAR 2
- BATCAR 3

Next:

- Set prism control to volume
- Set height to 0.004
- Set growth ratio to 1.3
- Set number of layers to 35

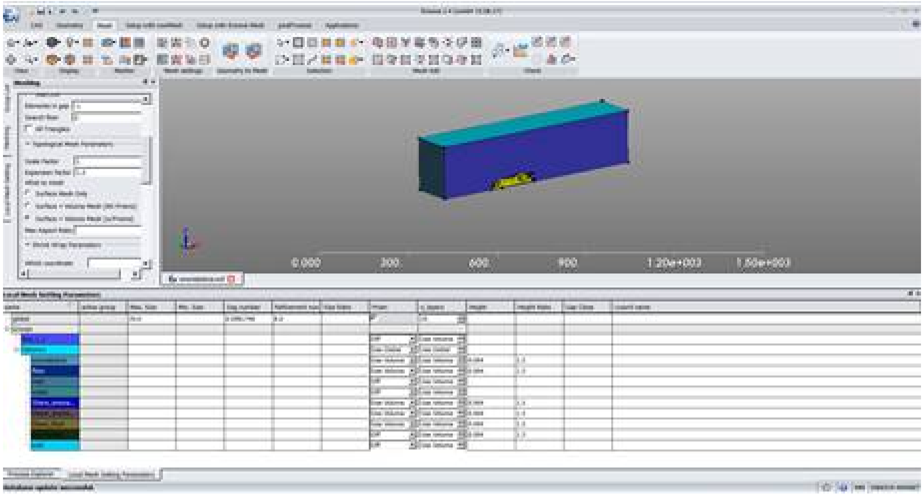
Press **Accept**. Open the mesh parameter settings spreadsheet and obtain a different representation of the model.



Volume Mesh with Prisms

Go back to the **Setup Meshing** icon and change the **What to Mesh** rule to **Surface + Volume Mesh with Prisms**. When asked, select **Yes** to clear the existing mesh.

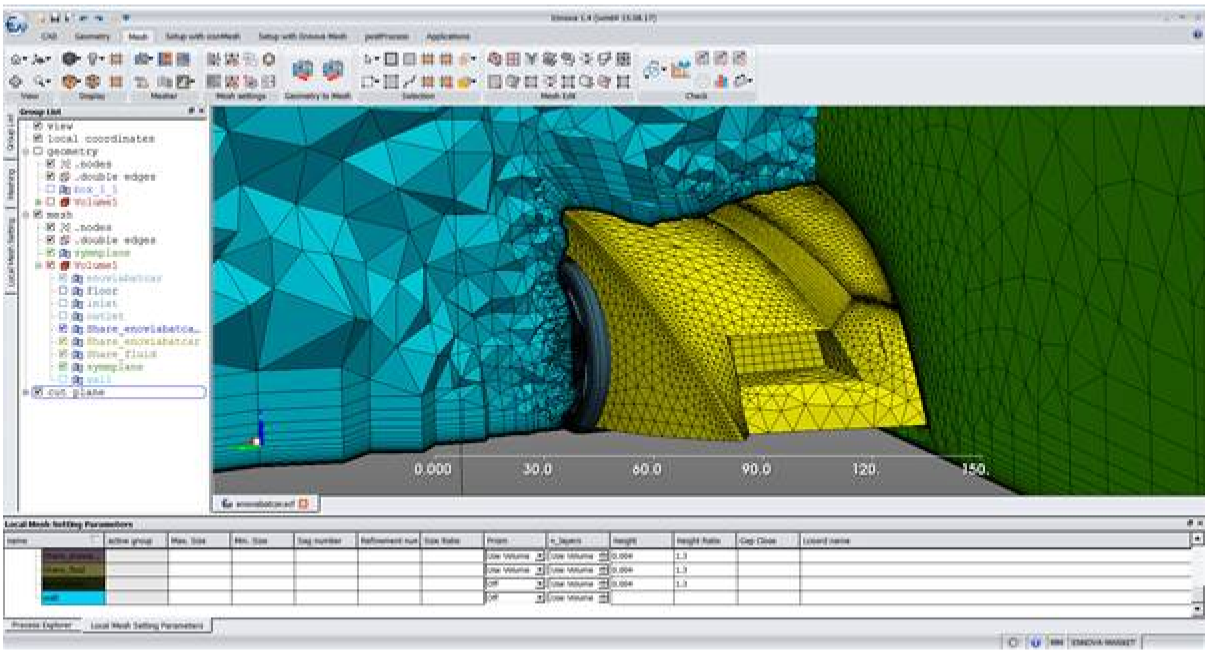
The complete mesh will be generated.



Mesh Diagnostics and View Slice

Check the mesh once it is complete. You can run the same mesh diagnostics as we used for the surface mesh earlier.

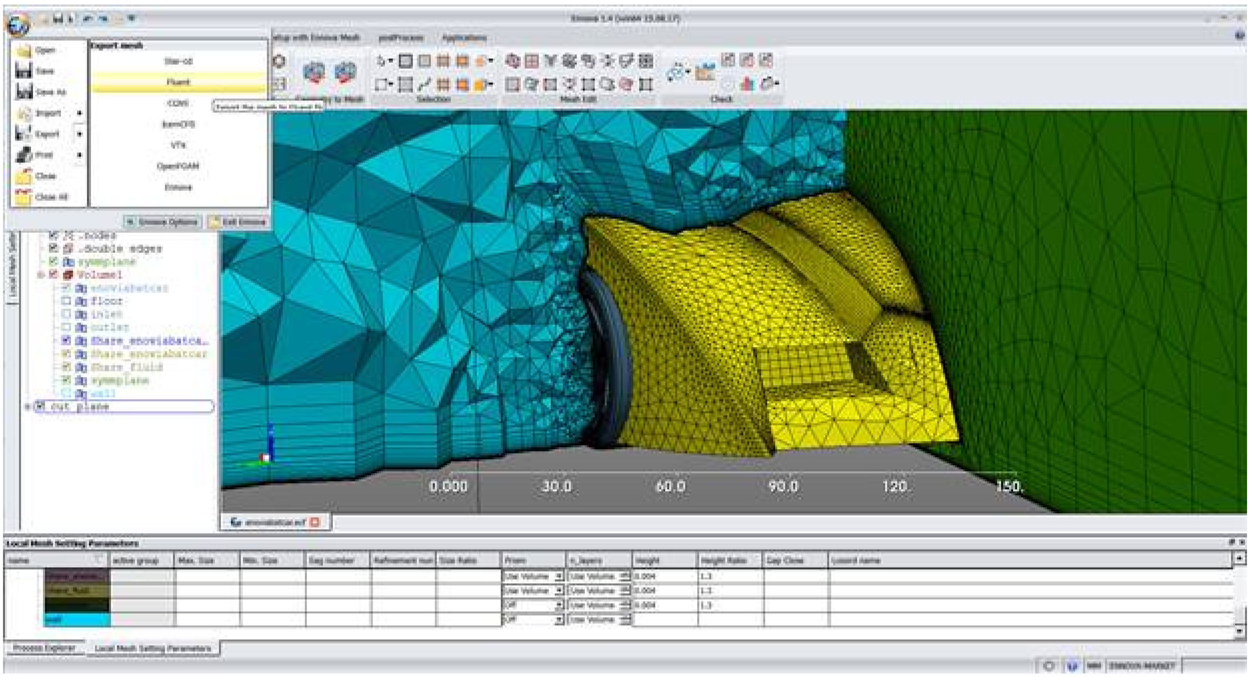
Use the Slice Tool to view the interior mesh.



Export to Fluent Format

Use the Ennova Icon to find the Export command. Choose Fluent Export.

The mesh is now complete.


Export mesh
Star-cd
Fluent
CGNS
OpenFOAM
0.000
30.0
60.0
90.0
120.
150.

DrivAer Car Tutorial

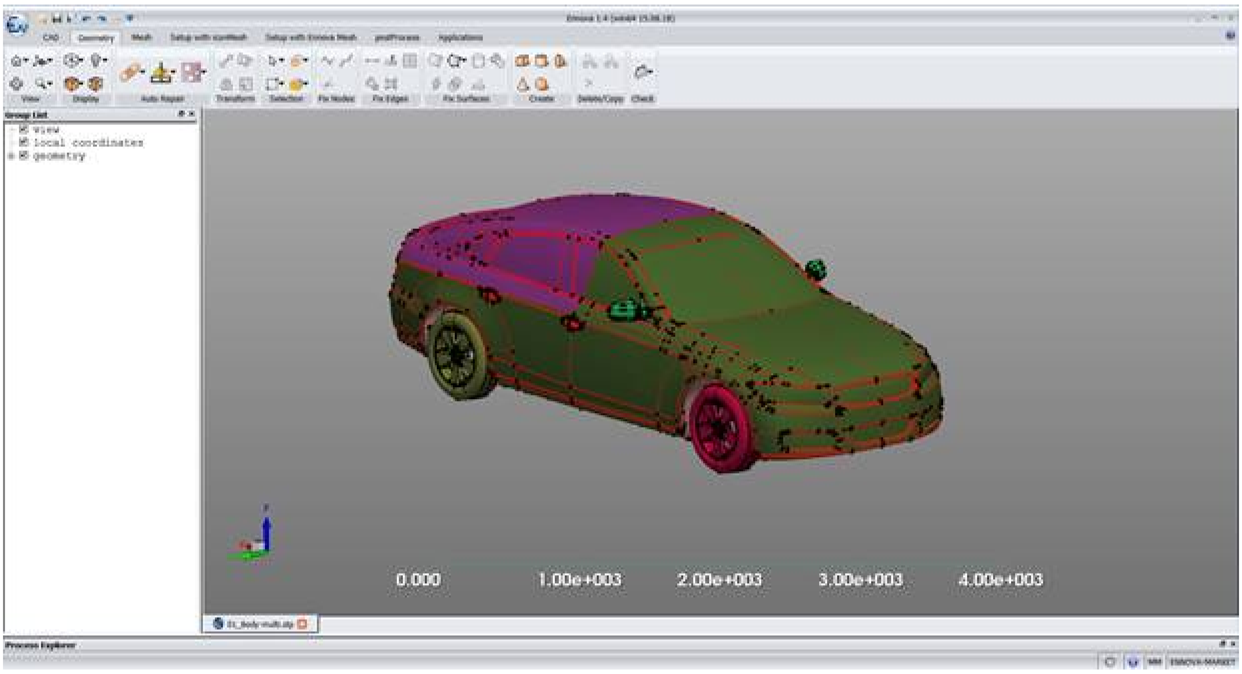
The DrivAer car is available after registering at this URL:
http://www.aer.mw.tum.de/en/research-groups/automotive/drivaer/

Once the download is complete, this tutorial uses the following parts:

- 01_Body.stp
- 02_Underbody_Detailed.stp
- 03_RearEnd_Fastback.stp
- 04_ExhaustSystem.stp
- 06_Wheels_Rear_Smooth.step
- 05_Wheels_Front_Smooth.step

The CAD Model

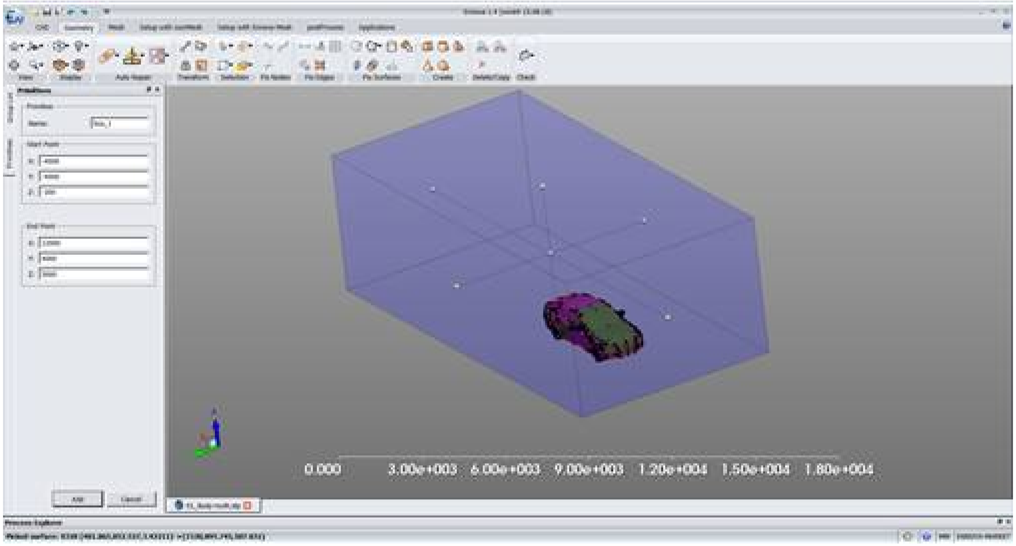
Loading the parts listed above will give the following model in Ennova:



CFD Domain

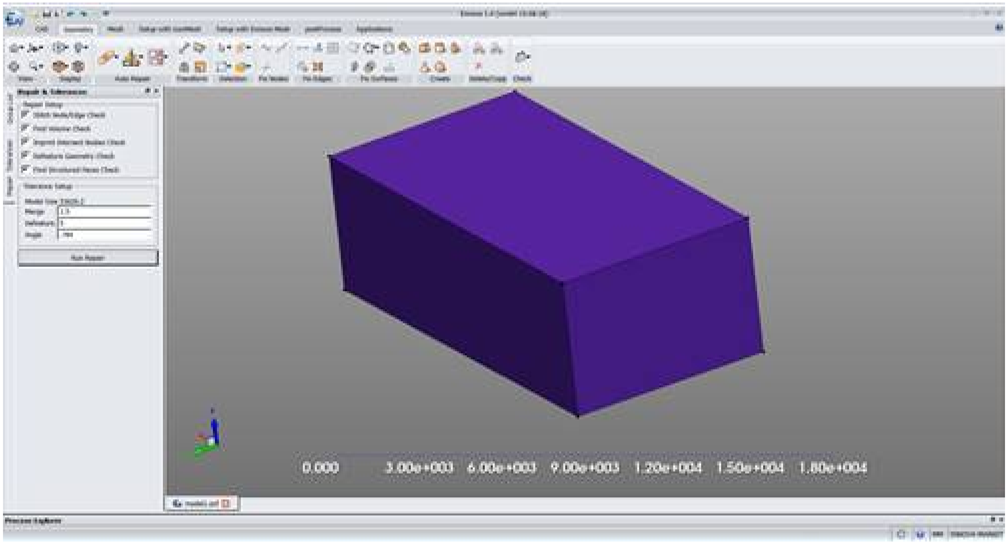
Add a box as CFD Domain.

Note: We will keep the domain small for the purposes of the tutorial, but a CFD domain of 1-2 car lengths upstream and 6-8 downstream would be a normal minimum.



Repair
Set repair tolerances listed below and Run Full Repair:

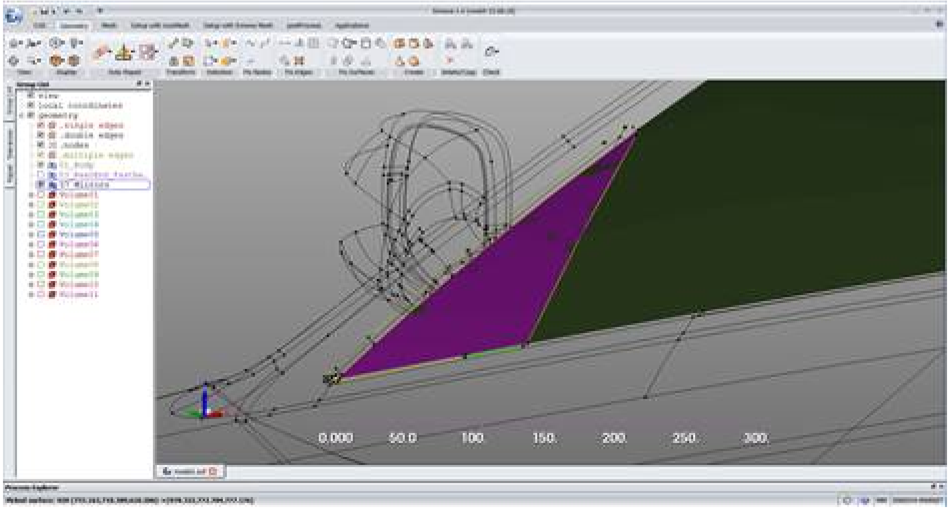
- Merge: 1.5
- Defeature: 0 (we will not defeature until later)
- Angle: 0.784



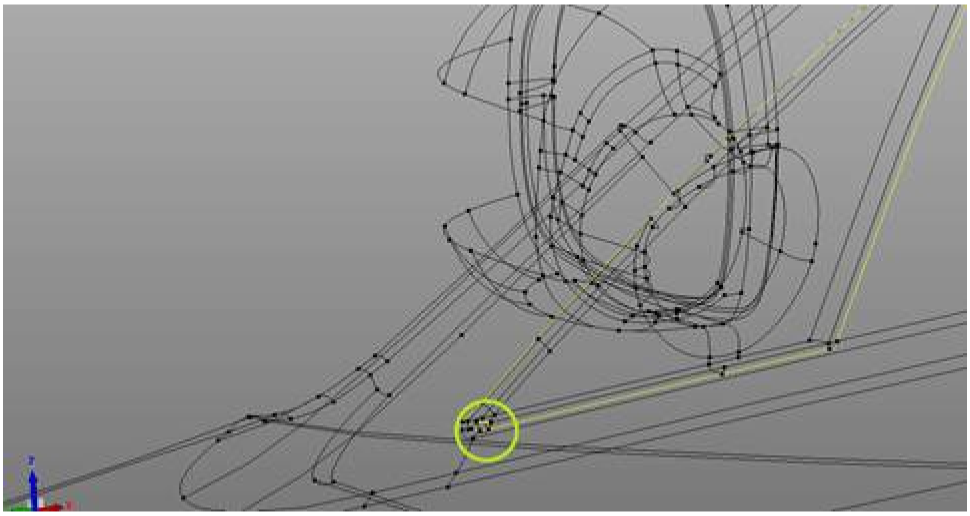
Further Manual Repair
Almost the entire repair of the car has occurred automatically. Ennova has largely been able to understand the full topology. Looking at the tree for hints we see the mirrors have not been successfully created as volumes.

On inspection there is a double face where the mirrors attach to the car. Like the exhaust, had we removed these surfaces before the merge they would have been treated correctly by the automatic routines. Sometimes it is easier to simply help the automatic repair ahead of time. Or just delete the surfaces.

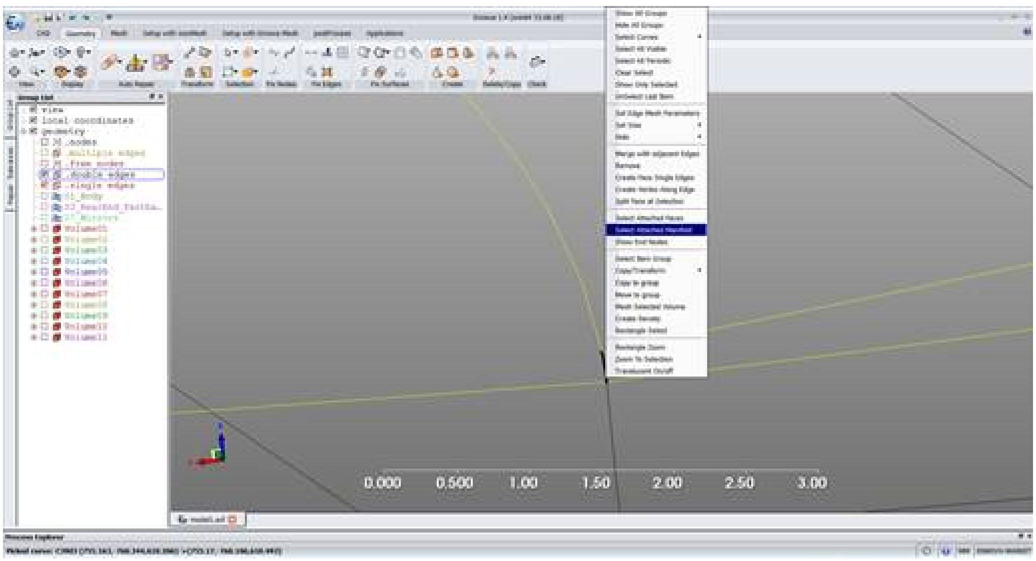
Use **RMB -> Remove** to delete the surfaces.



After deleting, notice that there is a tiny extra surface at the lower front as well. If you miss this Ennova indicates it by displaying it found “single edges “ in the tree.



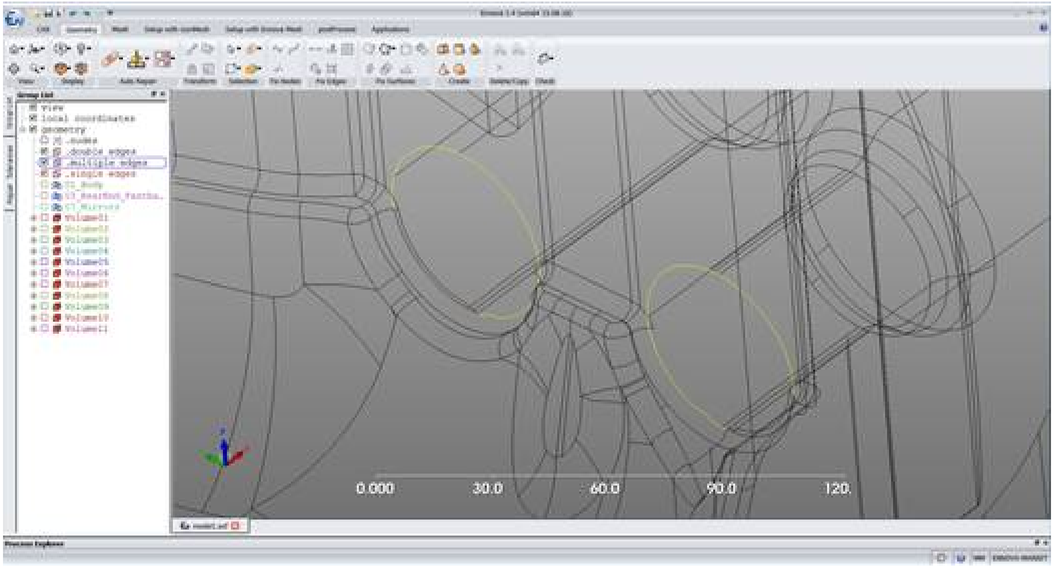
We do the same for the other mirror, but now that we know there are two surfaces to delete we can be more careful and use Select Attached Manifold to select all the surfaces at once. Often Select Attached Manifold is safer when there are large and small entities together.



Locate Single Edges

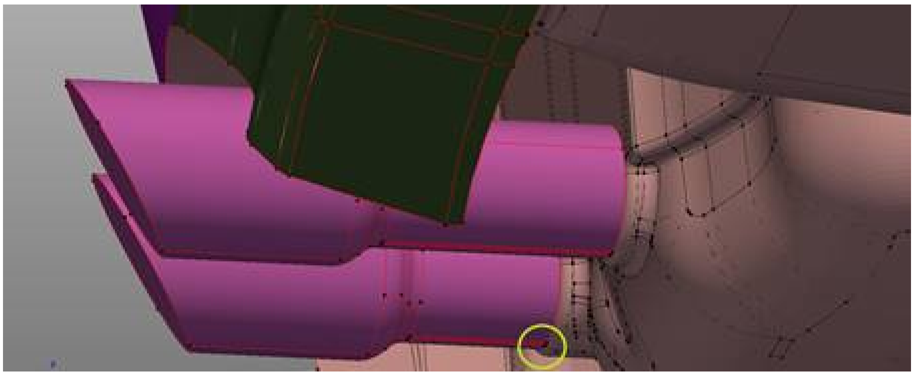
The next step is to locate the remaining single edges. For a closed volume that has no baffles we cannot have single edges.

The easiest way to do this is to turn off all nodes, edges, and surfaces and turn on only single edges. Find them in the Display by pressing the home icon. If they are still hard to see, use **RMB -> Select All Visible**. This will highlight them.

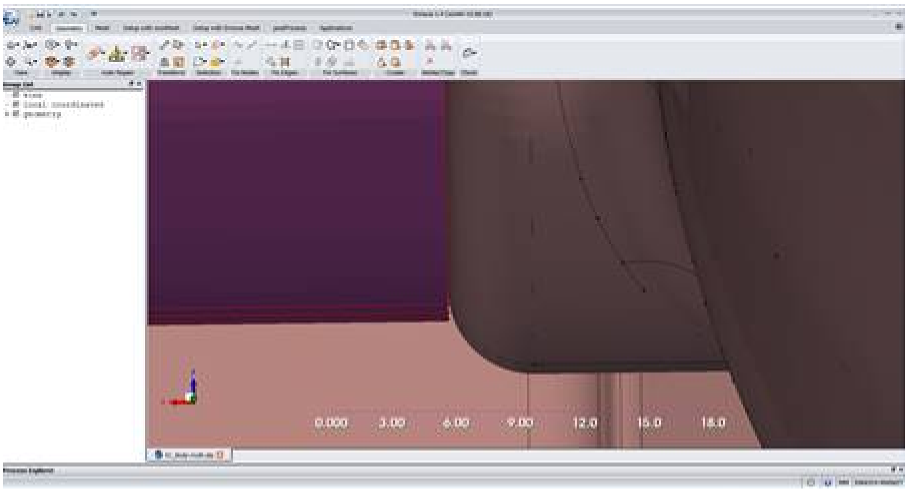


Exhaust Detail

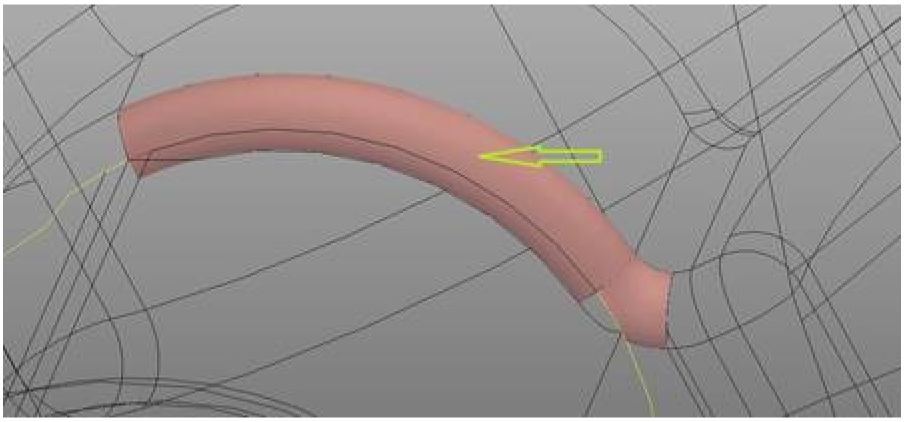
Once selected, we see an issue in the exhaust at the rear of the car.



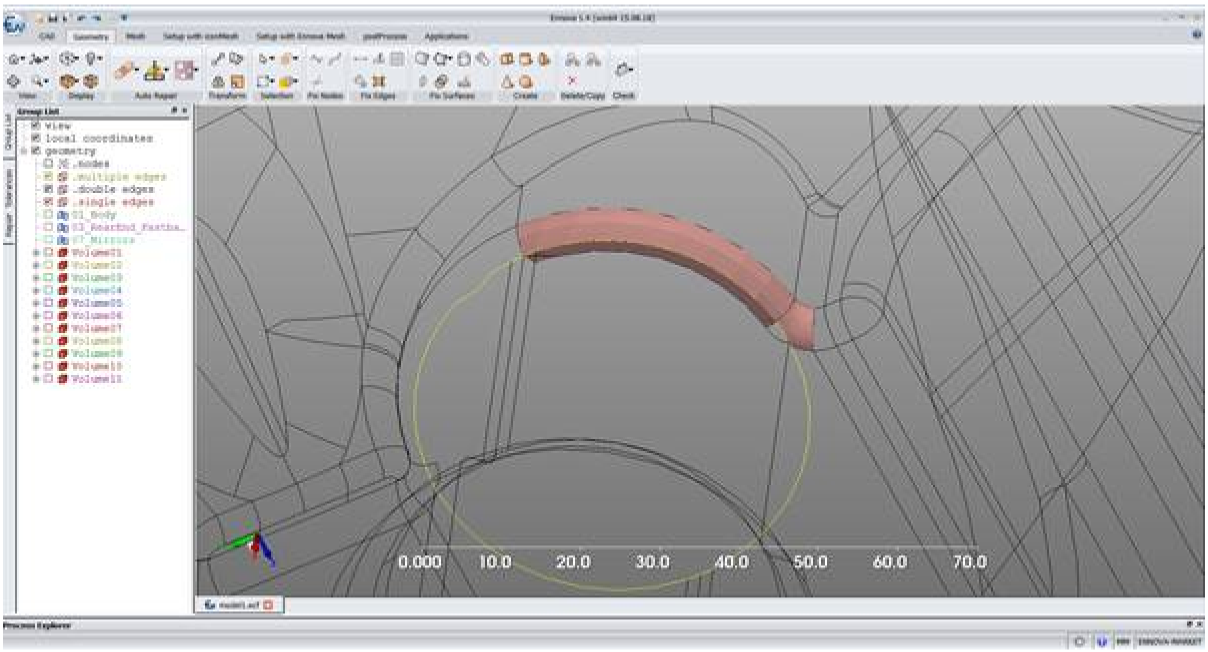
After inspection we see that on both exhaust pipes there is not an intersection with the car muffler. We need to manually split these surfaces.



The green arrow indicates the surface that needs splitting. Note both the Black and the Red edges need to be used for splitting.



Highlight the correct surface and edge and use **RMB -> Split Face at Edge** for both exhausts.



The yellow edge is now a closed loop. Rather than look for Red edges (single) we could also have searched the model for open Yellow edges indicating a problem.

Fix both exhausts.

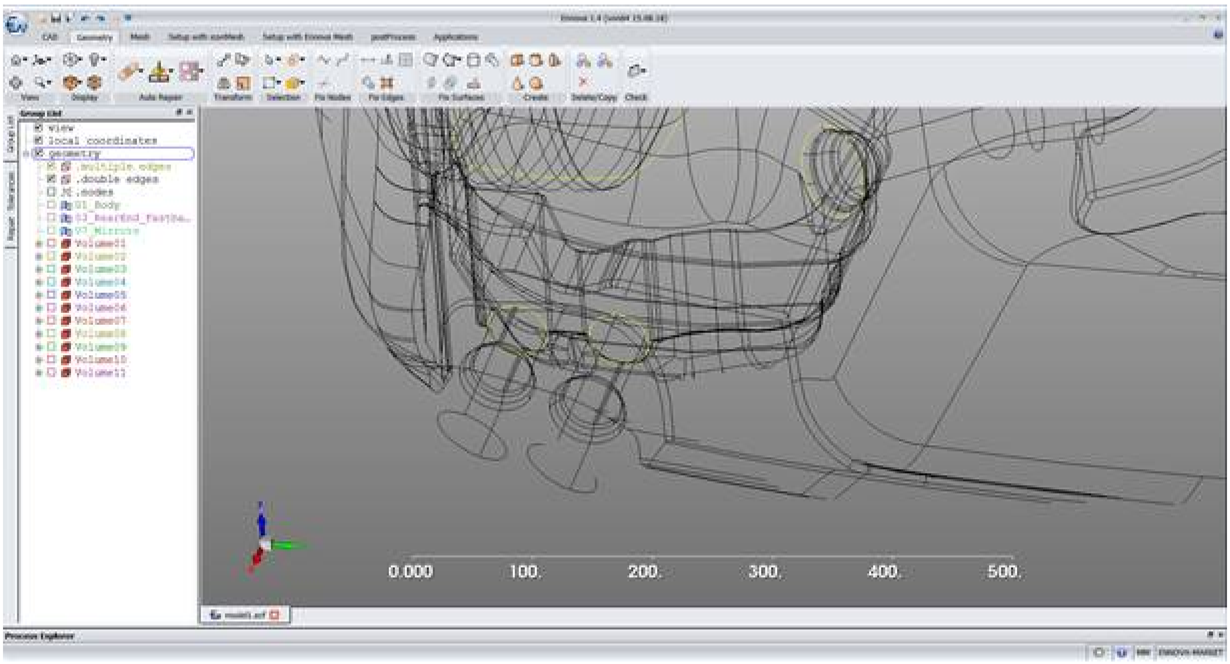
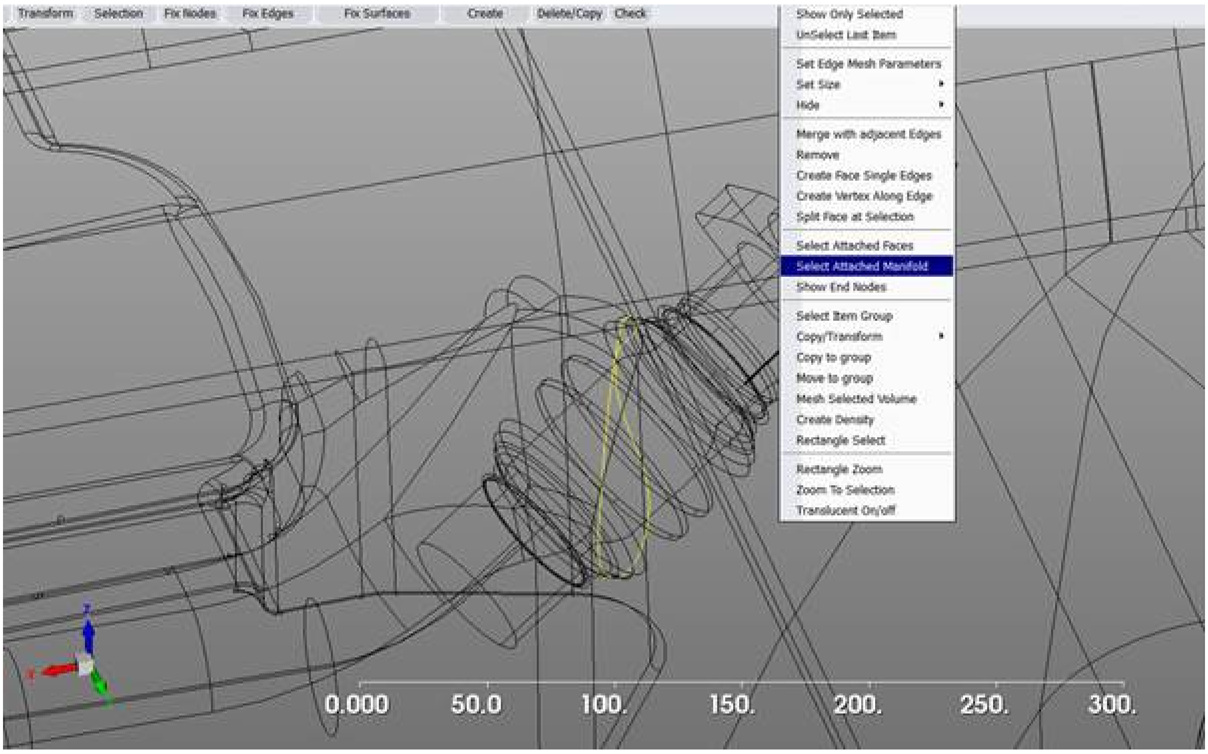
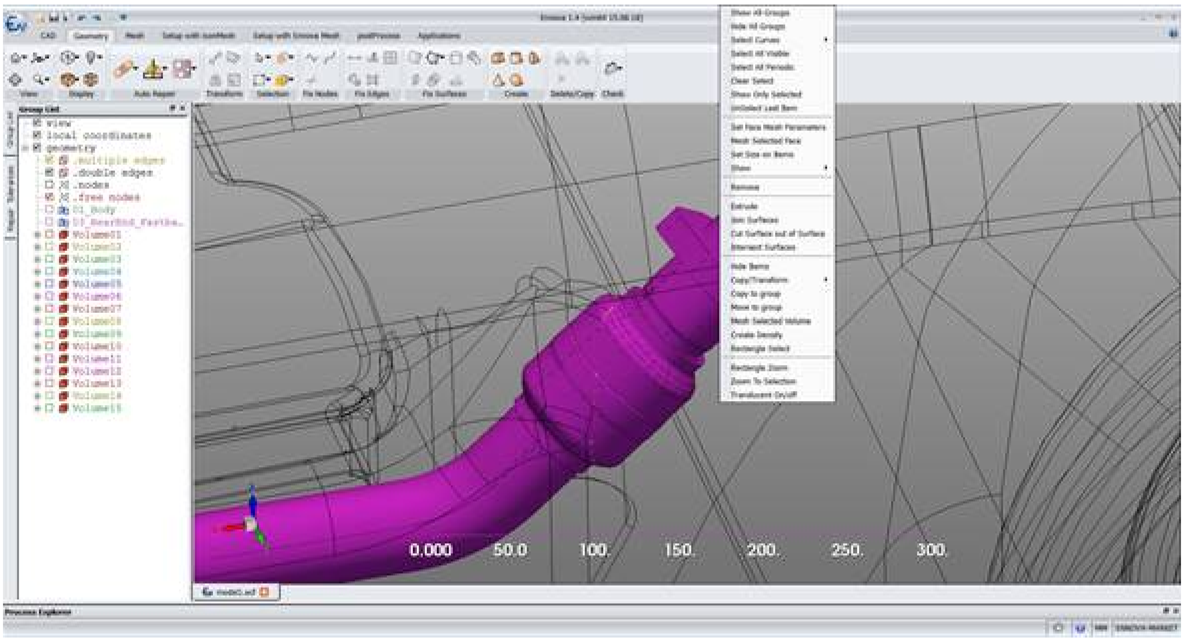


Fig. Rear Exhaust After Fix

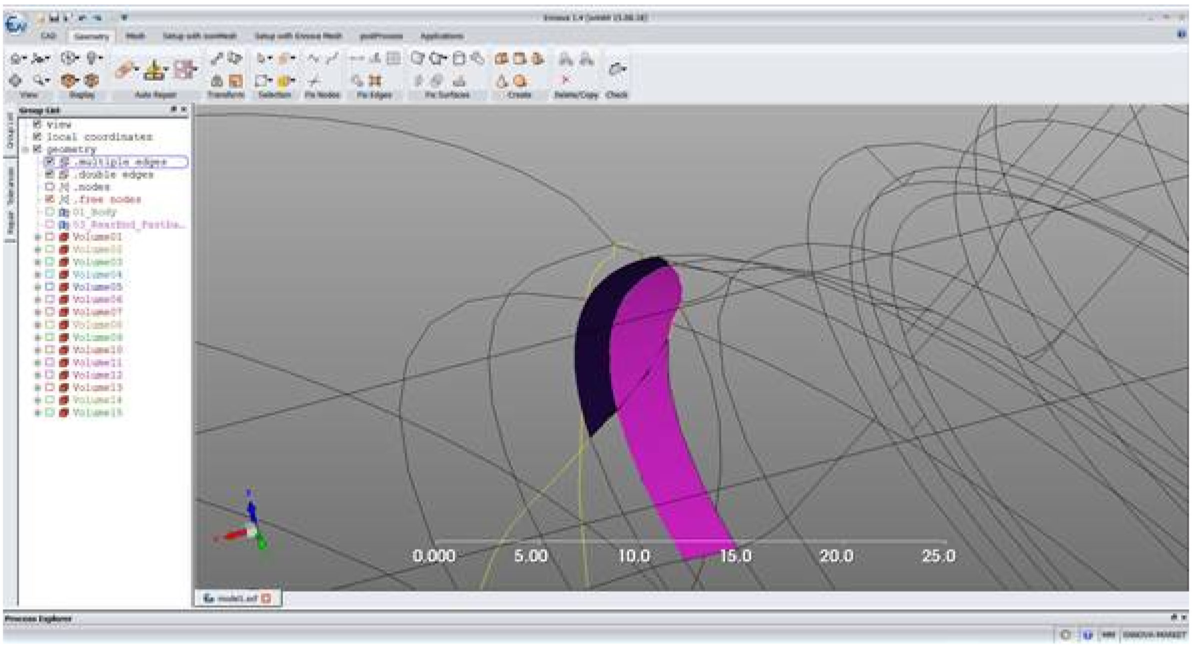
There is still one more open yellow loop in the model. It is harder to find, as it is a very small gap. One way to research a model is pick a black edge that is on a manifold that should be able to be displayed easy. For example the inboard end of the exhaust header should be one manifold for just until it crosses the body.

Selecting the manifold highlights the whole exhaust, so this yellow loop is leaking.

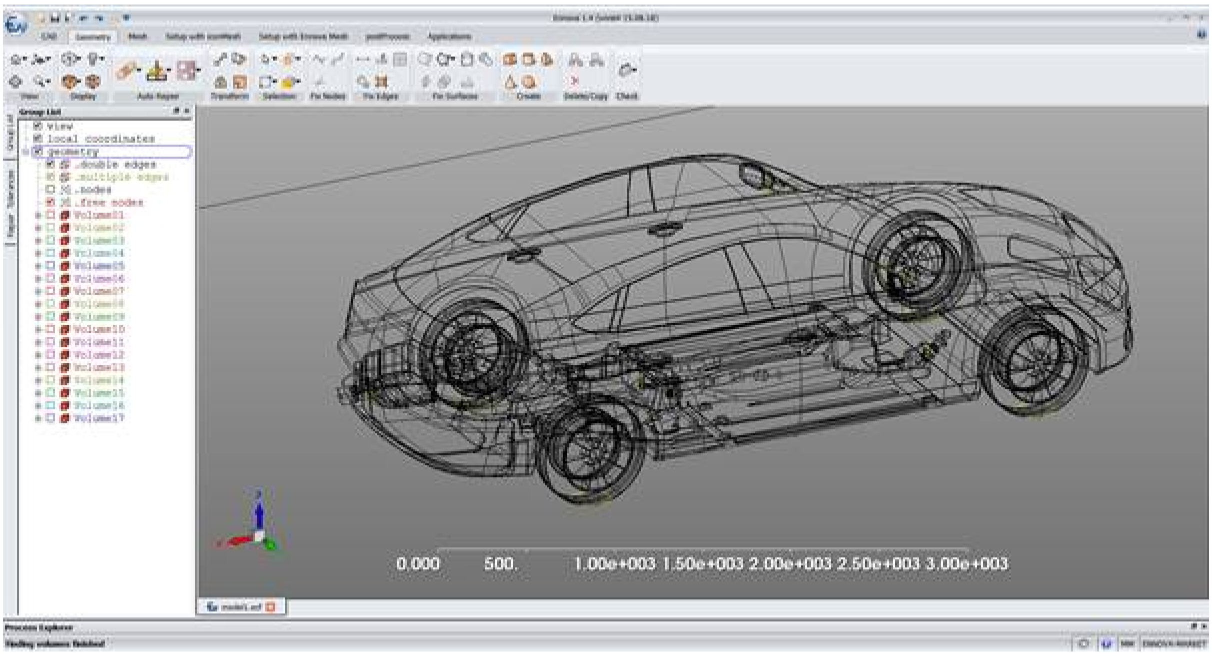




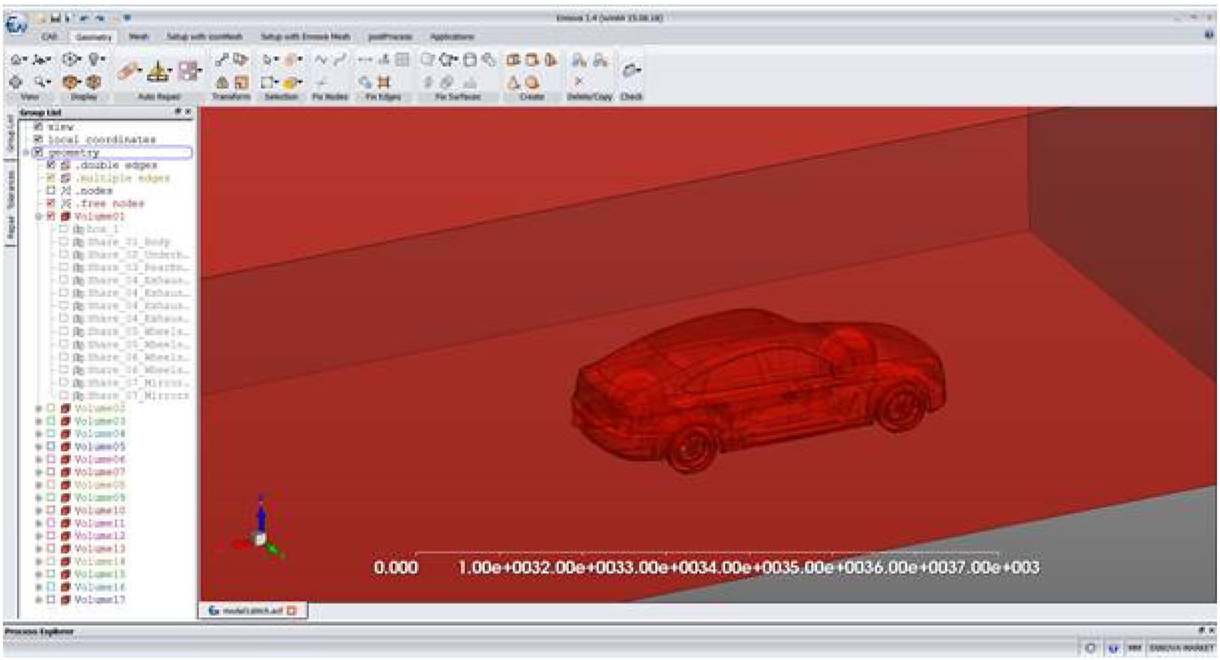
On close inspection there are four surfaces intersecting here and one of these did not split.   Use **RMB -> Split Face at Edge** to fix as usual.



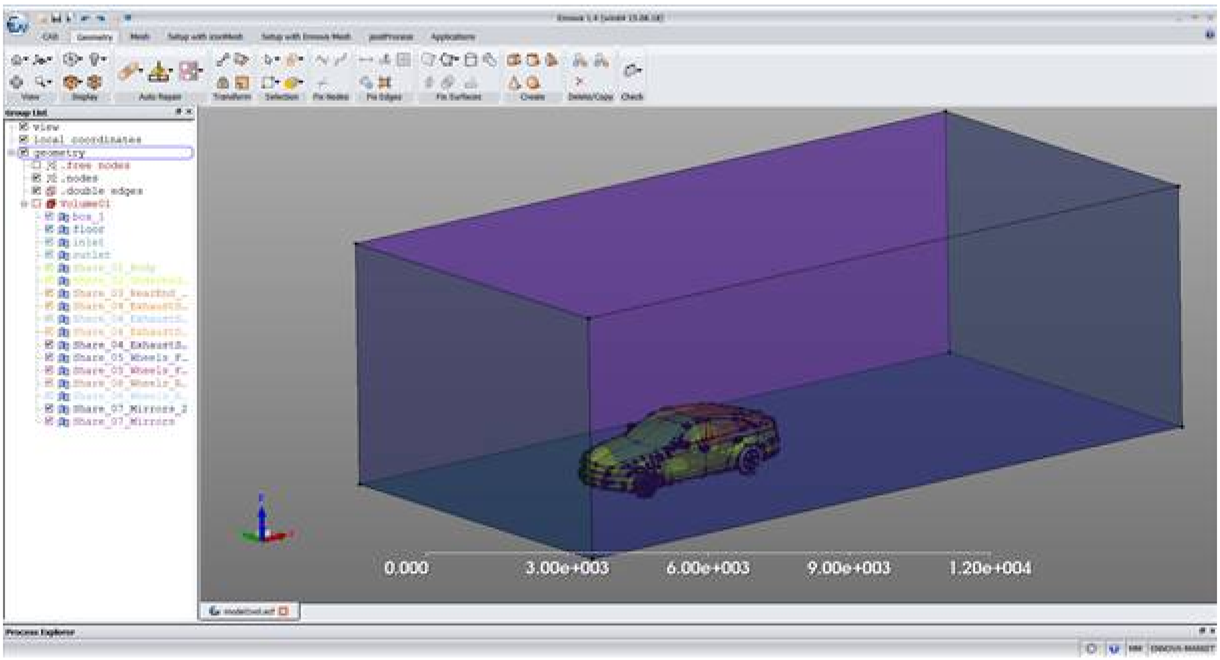
The model is now complete. Use the Find Volume command to reveal 17 volumes.



Volume 1 is the CFD Domain we want. Use **RMB -> Delete Group and Contents** on all other volumes.



Select and Move boundaries to appropriate groups.



The model is now ready for meshing.

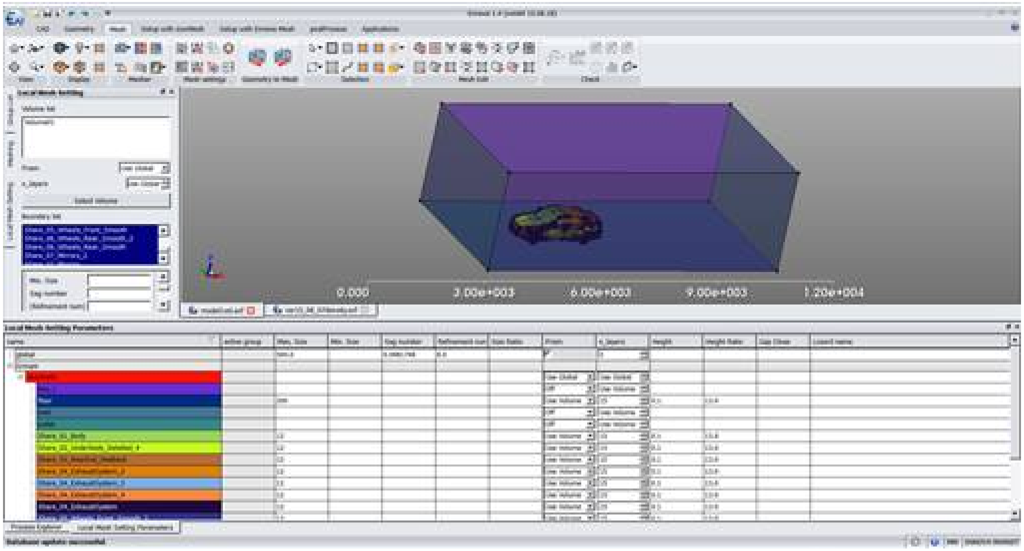
Mesh Control

Set maximum sizes:

- Car 12 mm
- Wall 500 mm
- Floor 200 mm

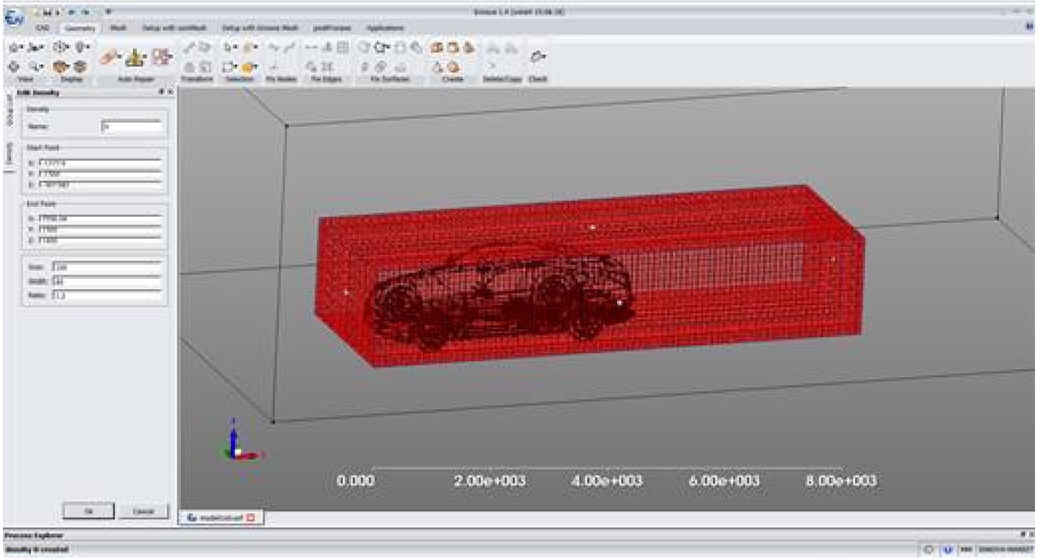
Set Blayer height to 0.1 mm with 15 layers.

Use the Mesh parameter spreadsheet and dropdown box.



Basic Mesh Parameters

Create Density Box around Car and Wake



Defeature Model

Now the model is a valid volume. Set the defeature parameter to 5 mm and run the Defeature command. This will remove any CAD artifact that did not mesh well.

Note: We are only defeaturing the topology of the mesh. The underlying CAD geometry is unaltered.

A final surface mesh x, y, z point always lies exactly on the original CAD geometry.

Surface Mesh – Diagnostics 1

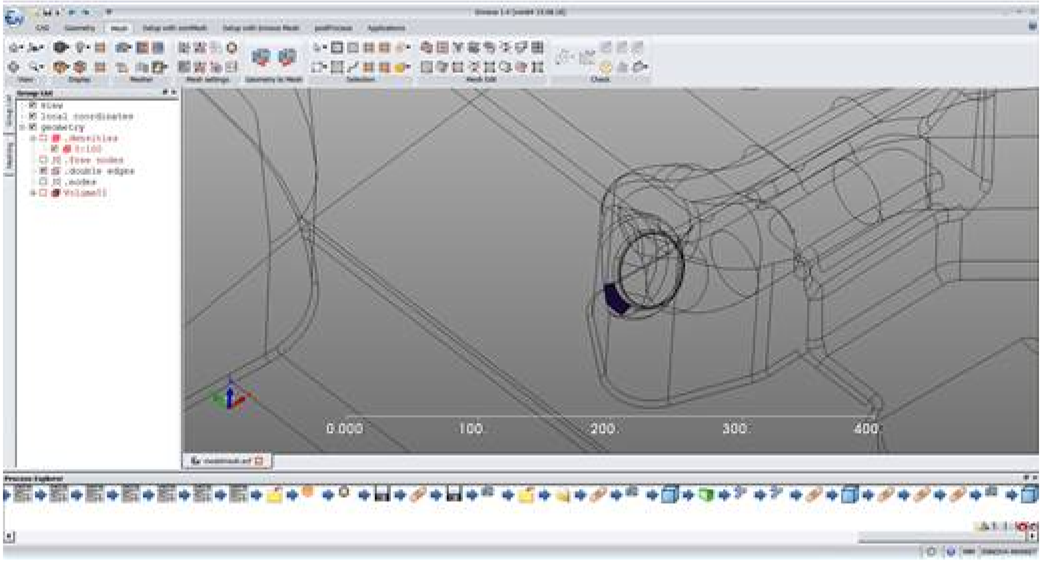
At this stage, try a surface mesh. Ennova will come back reporting that a partial mesh has been created and ask whether you want to load it. Answer No to this, and No to viewing the unmeshed faces. Instead, look in the check mesh and there will be an unhappy face. Click on this and it will take you to the model face that could not mesh.

There is no tutorial for this as it is a sliver surface, extremely thin and hard to view. Track it down by turning everything else off except the smiley face.

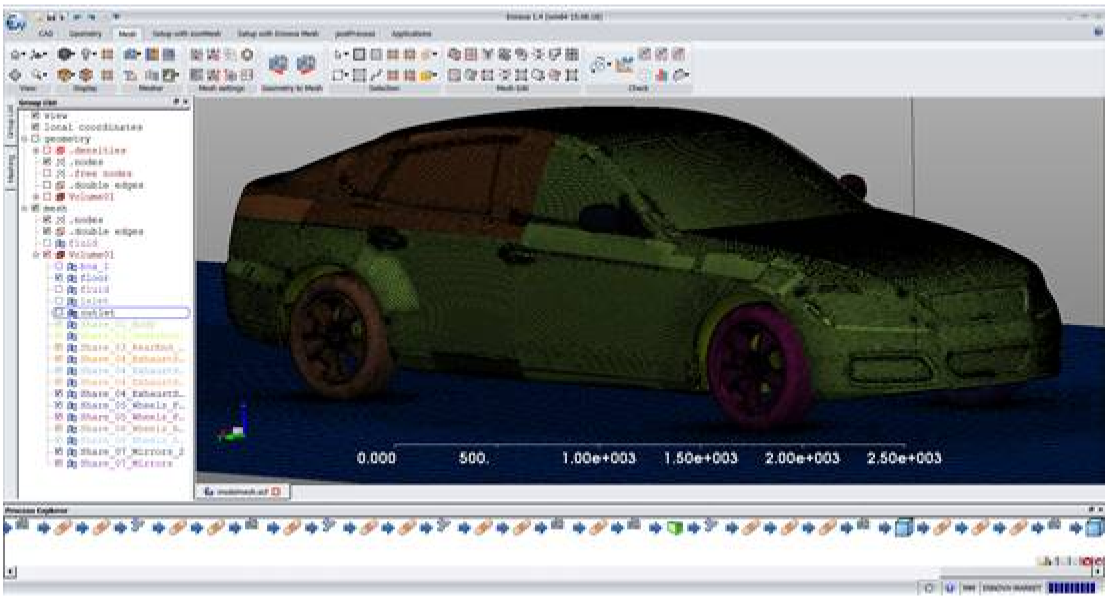
Delete the face and there are two red edges on the left close together. Merge those nodes together using Merge Adjacent Edges. Remesh again.

Use the unhappy face and the Home icon to find the next unmeshed face. It is easier to see and is on the exhaust.

As this is in a non-critical location, simply removing the face and using **RMB -> Create Face at Single Edges** will fix the issue. The face itself is overlapped and not trimmed correctly.



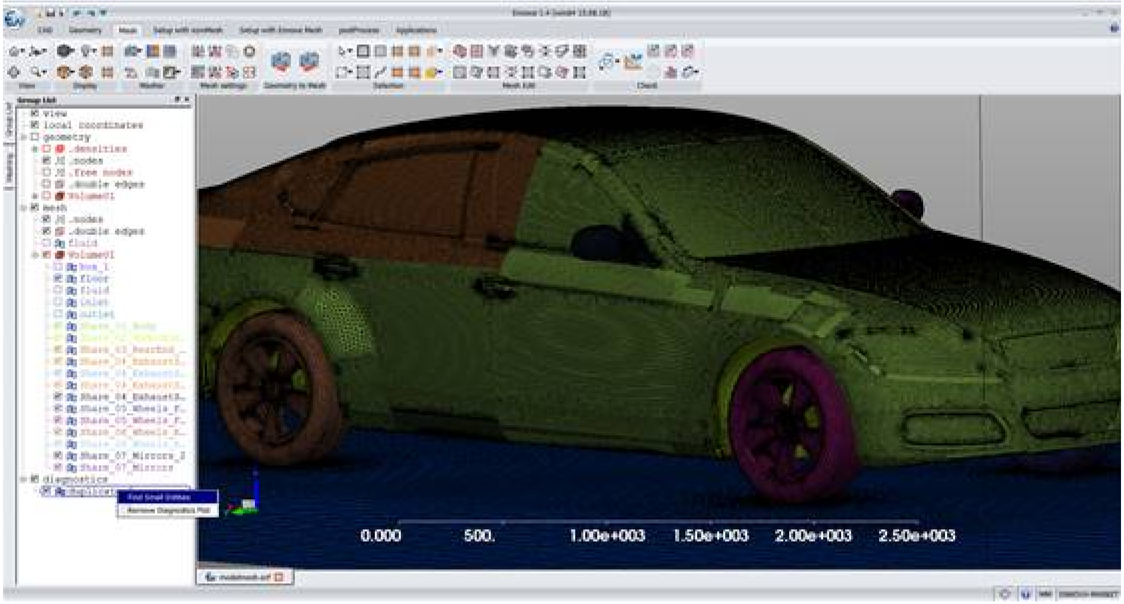
Repeat this procedure again to find one more sliver surface. Then a complete surface mesh can be loaded.



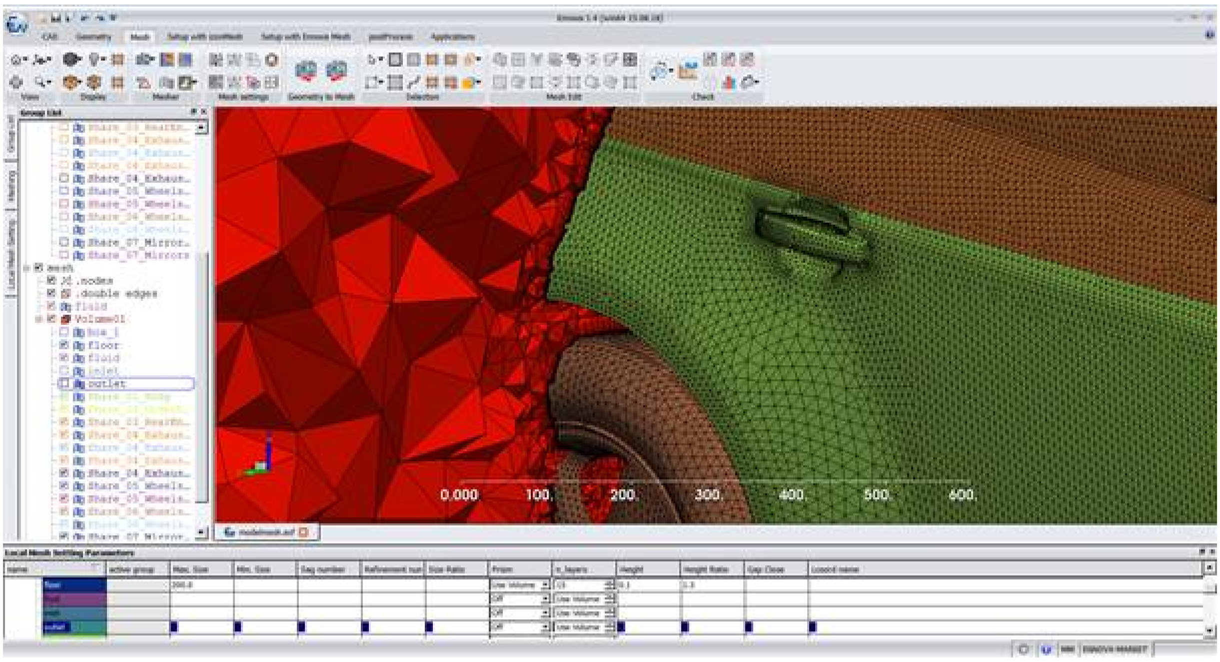
Surface Mesh Diagnostics

After running the mesh diagnostics we find there are some duplicate faces. Use the find small entity tool to find them.

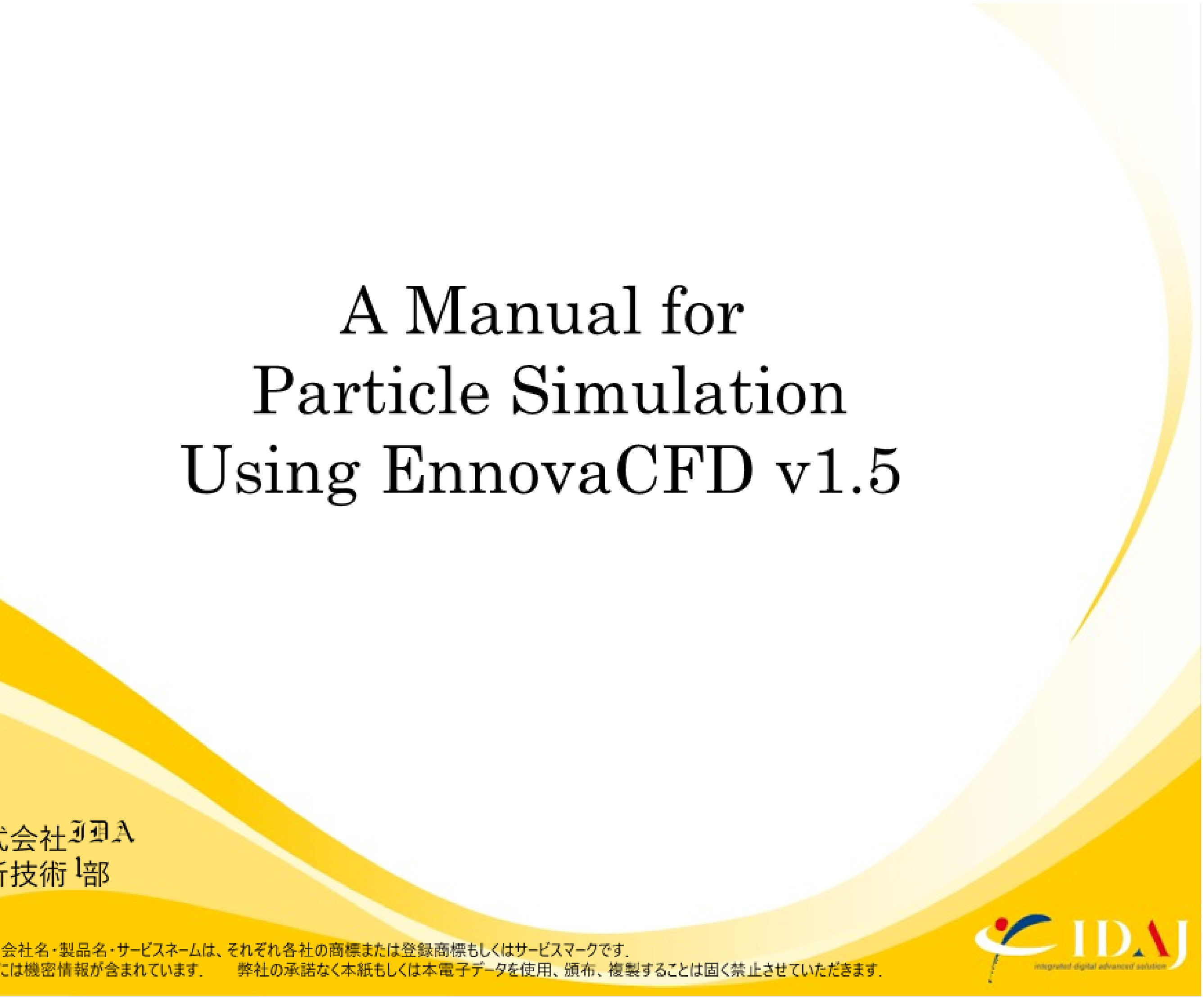
In this case an edge is not meshing/projecting properly. Simply split the edge with a vertex and re-mesh. A valid mesh will result.



Final Volume Mesh with Prisms



At this stage a valid mesh should result. Similar to before mesh diagnostics can be used to verify the quality of the mesh. Using the Ennova button the mesh export dialogue box can be used to export the mesh to the desired format. This concludes the tutorial.


A Manual for
Particle Simulation
Using EnnovaCFD v1.5
会社IDA
技術1部
会社名・製品名・サービスネームは、それぞれ各社の商標または登録商標もしくはサービスマークです.
には機密情報が含まれています.　　弊社の承諾なく本紙もしくは本電子データを使用、頒布、複製することは固く禁止させていただきます.
IDAJ
integrated digital advanced solution

Contents

Settings for steady→unsteady particle simulation

1. Importing the data
2. Setting the iconCFD solver
3. Running the simulation
4. PostProcessing

Settings for unsteady, coupled fluid-solid particle simulation

1. Importing the data
2. Setting the icoCFD solver
3. Execution and postProcessing

he targeted version:

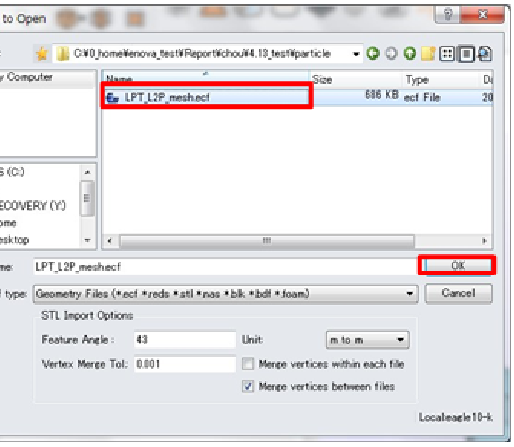
ennovaCFD1.5

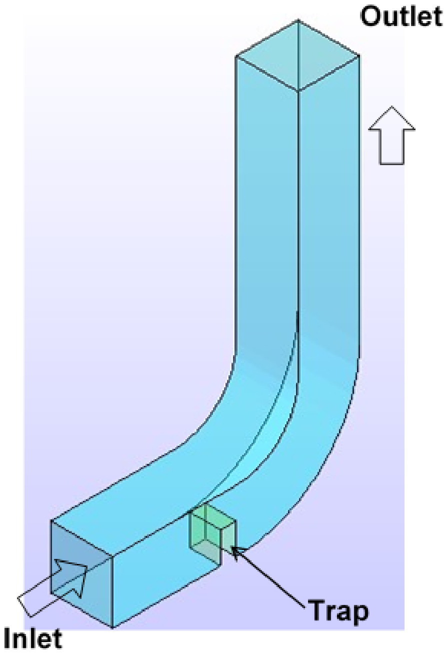
iconCFD3.2.11

Settings for steady→unsteady particle
mulations

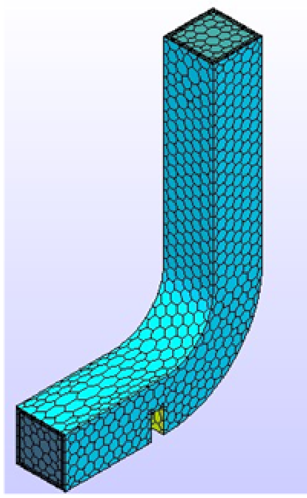
1-1. Importing the

Use the LPT_L2P.ecf in the sample data. It has already been meshed with ennovaCFD.
Import LPT_L2P.ecf



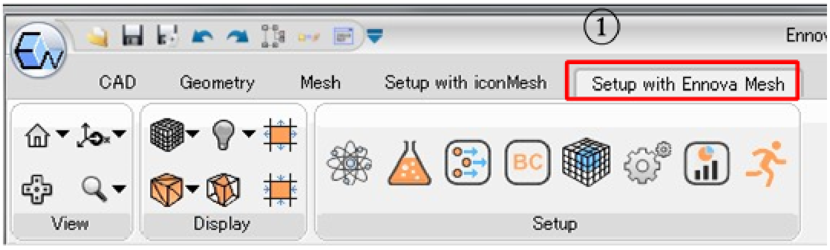


Polyhedral mesh



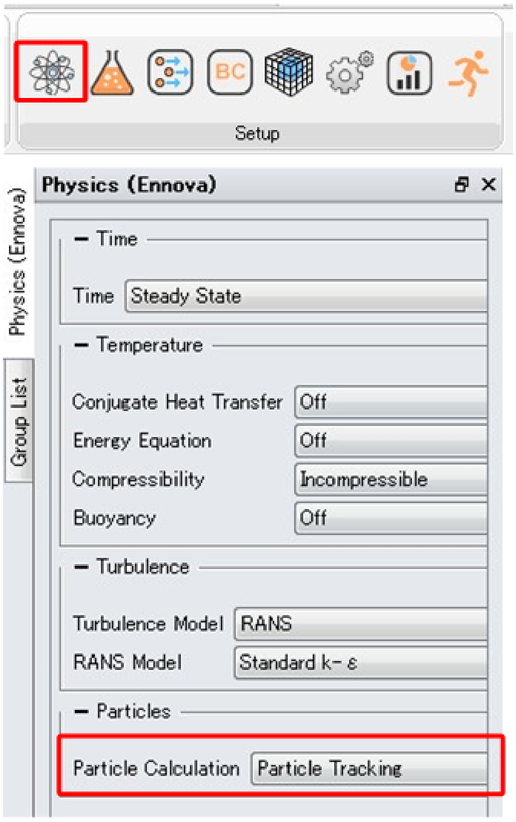
1-2 Setting the iconCFD

Since the polyhedral mesh has already been set up by ennovaCFD, we could proceed to set up the solver with Setup with Ennova Mesh.



1-2 Setting the iconCFD

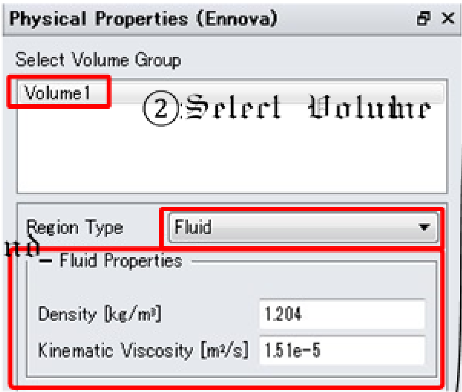
In the current case, after carrying out the continuous incompressible, steady state simulation, we will use the results to perform the unsteady simulations (discrete phase). In the Physics panel, we will set up for physics of the continuous phase (steady/unsteady, compressible/incompressible, temperature, turbulence) and particles.
Set the solver as follows



- Time :Steady State
- Conjugate Heat Transfer :Off
- Energy Equation :Off
- Compressibility :Incompressible
- Buoyancy :Off
- Turbulence Model :RANS
- RANS Model :Standard k-ε
- Particle Calculation: Particle Tracking

1-2 Setting the ironCFD

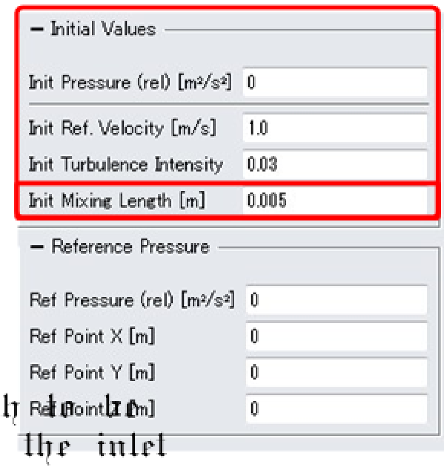
In Physical Properties panel we set the physical properties for the continuous phase. In this case, we will use the physical properties of air.



②Select Volume

...ct Fluid and
...e default
...properties

④Initial (pressuer, turbulence)

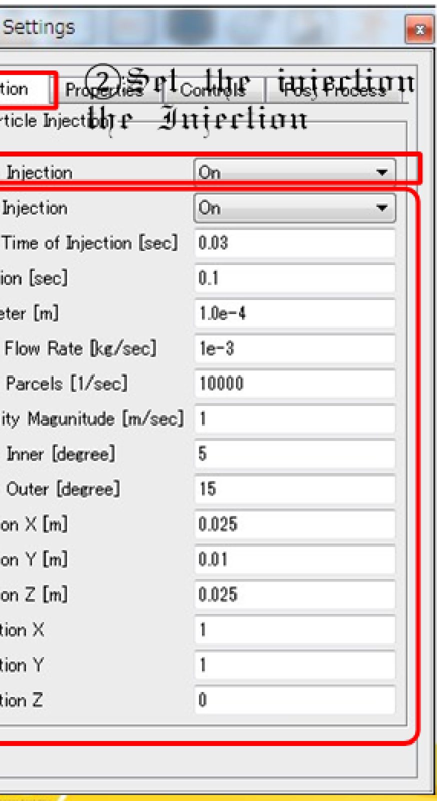


⑤ Set the Init Mixing Length to ... of the characteristic length of the inlet (in the current case0.005)

1-2 Setting the iconCFD

articles Settings

- Explanation of the Injection tab



② Set the injection method of the particle at ① the Injection

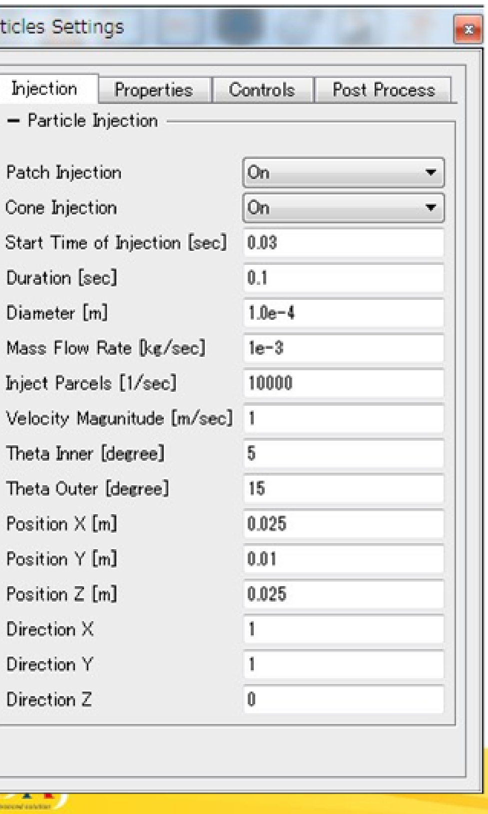
③: When injecting the particle from the bo select Patch Injection On. Detailed information will be set at the Boundary (p.14

④: In the case of Cone shape spray the parameters are set as

1-2 Setting the iconCFD

articles Settings

- Explanation of the Injection tab (Contd)

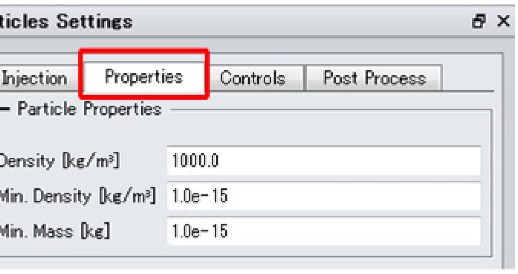


Start Time of Injection : start time of injection[s]
Duration : Duration of the spray[s]
Diameter : diameter of the particle[m]
Mass Flow Rate : mass per second[kg/s]
Inject Parcels : number per second[1/s]
Velocity Magnitude : velocity of the particle[m/s]
Theta Inner/Outer : angle of the cone [deg] spray in between the Inner~Outer angles
Position (XYZ) : position of spray point[m]
Direction (XYZ) : direction of the spray

1-2 Setting the iconCFD

articles Settings

- Explanation of the Properties tab

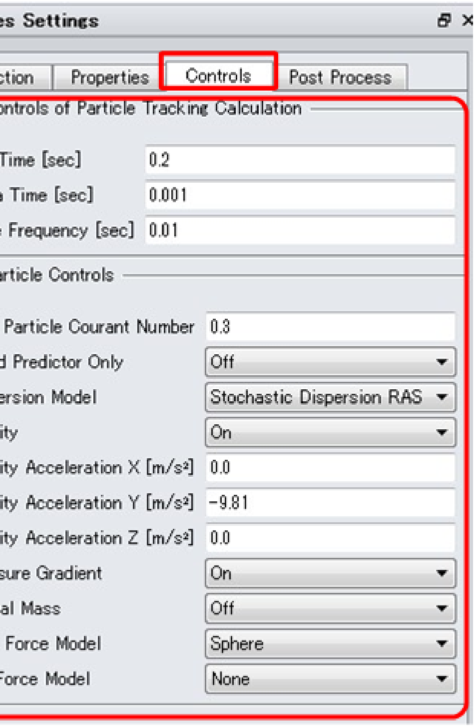


Density : density of the particle [kg/m3]
Min Density : minimum density of the continuous phase at the location of particle [kg/m3]
Min Mass : the minimum weight of the particle [kg]

1-2 Setting the iconCFD

articles Settings

- Explanation of the Controls tab

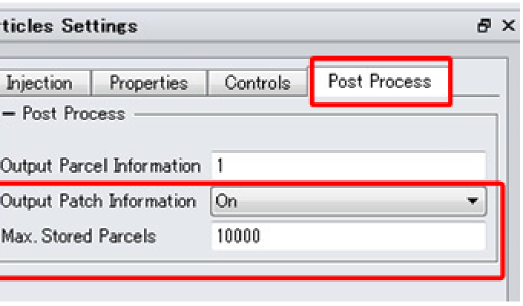


End Time : End time of simulation [s]
Delta Time : time steap [s]
Write Frequency : output interval [s]
Max Particle Courant Number : Maximum courant number
Could Predictor Only : When it's ON, the particle calculation will be carried out for each timestep.
Dispersion Model : particle dispersion model
Gravity : Gravity ON/OFF
Gravity Acceleration (XYZ) : Direction of gravity
Pressure Gradient : Force from the pressure gradient ON/OFF
Virtual Mass : Force from the virtual mass ON/OFF
Drag Force Model : drag force model
Lift Force Model : lift force model

1-2 Setting the iconCFD

articles Settings

- Explanation of the Post Process tab



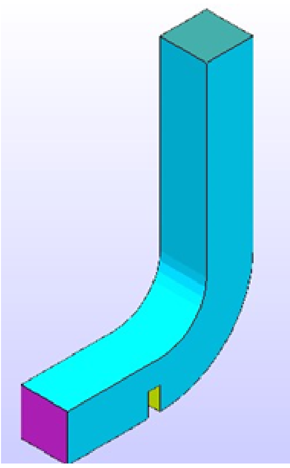
Output Parcel Information : output parcel information of the particle
Output Patch Information : output of the patch information ON/OFF
Max Stored parcels : maximum stored particles

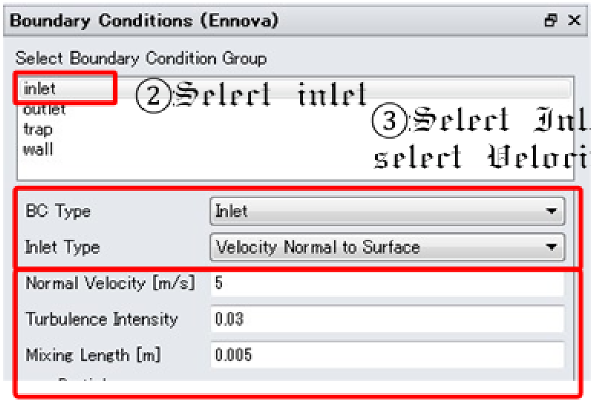
1-2 Setting the iconCFD

Boundary Conditions

- Set the boundary condition of the continuous phase at the inlet.

①





②Select inlet

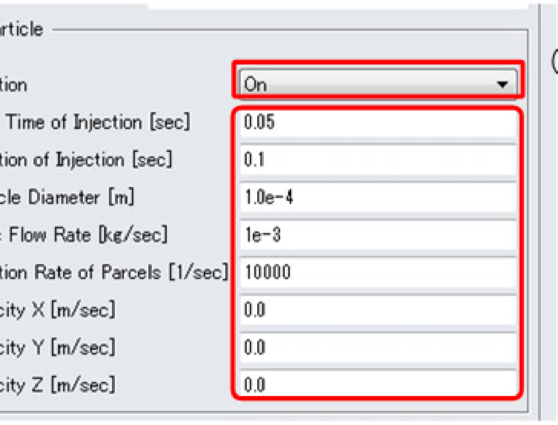
③Select Inlet at BC type, select Velocity Normal to surfa

④Set the velocity and the turbulence properties.

1-2 Setting the iconCFD

Boundary Conditions

- Next, set the Patch Injection of the particle at the inlet.

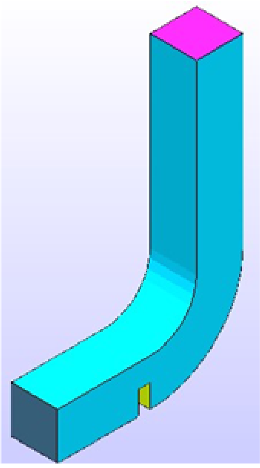


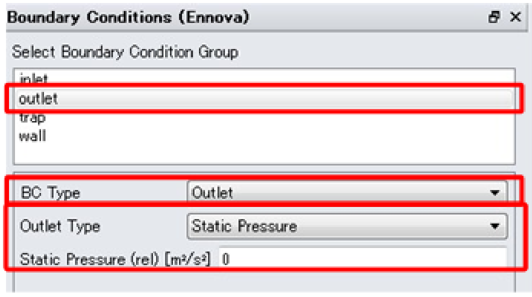
①Injection:

1-2 Setting the iconCFD

Boundary Conditions

- Set the boundary condition of the continuous phase of the outlet.





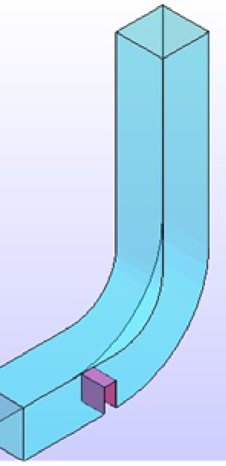
①Select outlet

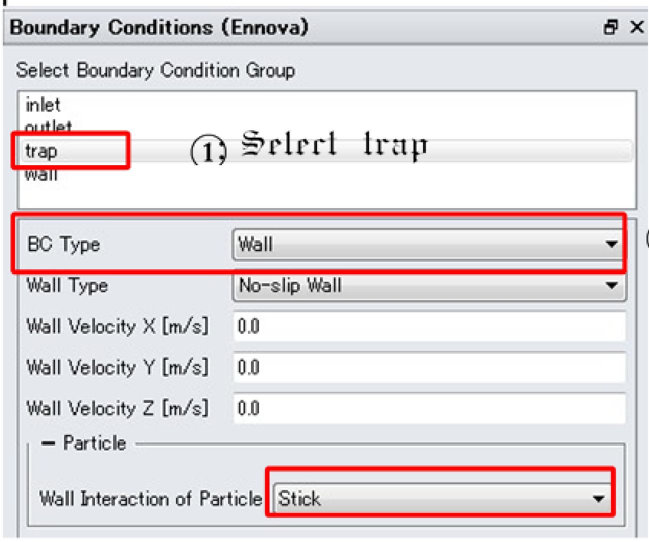
②Select BC type at outlet, and select Static Pressure.

1-2 Setting the iconCFD

Boundary Conditions

- Set the boundary condition of the continuous phase and the particle for the Trap.





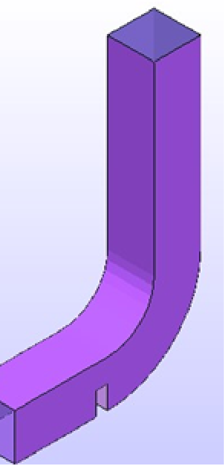
② Select Wall at BC Type (Default)

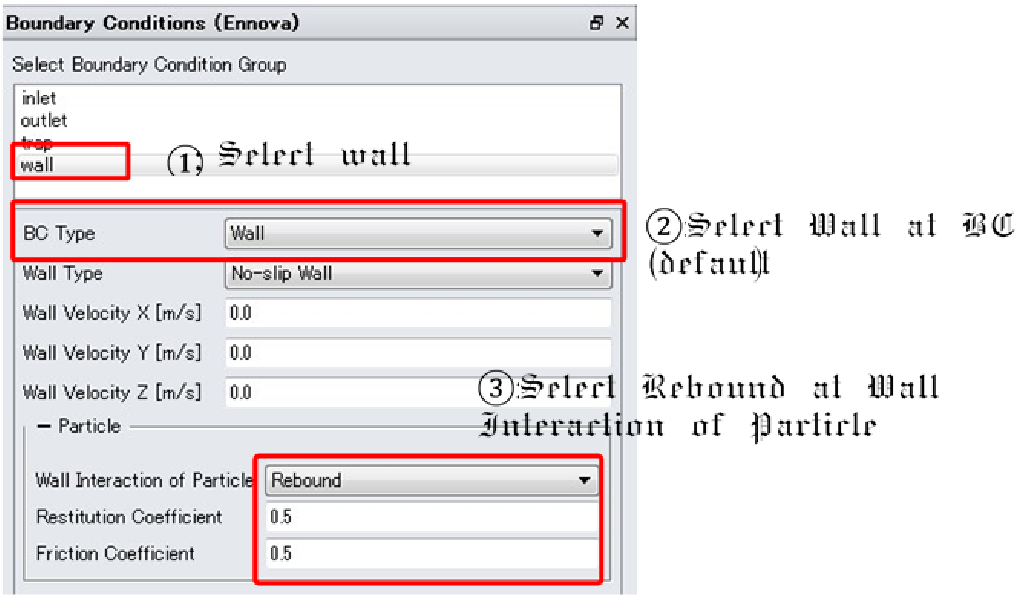
③ Select Stick at Wall Interaction of Particle

1-2 Setting the iconCFD

Boundary Conditions

- Set the boundary condition of the continuous phase for the wall and the particle condition on the wall.





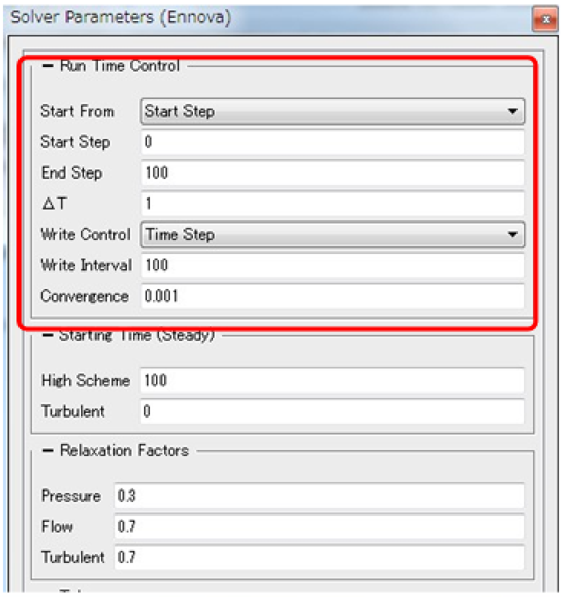
vp= (1-Firction Coef)vpt – (Restituion Coef)vpn

Vp: Velocity vector of the particle after rebounding
Vpn: Normal velocity vector of the particle before rebounding
Vpt: Tangential velocity vector of the particle before rebounding

1-2 Setting the ironCFD

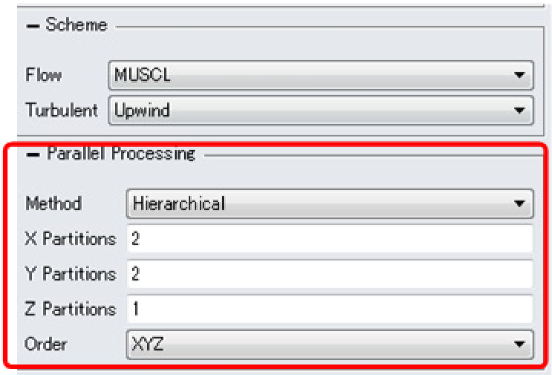
Set the solver for the continuous phase at Solver Parameters panel.

- Start Step/End Step
- ΔT
  - Time interval
- Write Control
- Write Interval
- Convergence
  - The criterion of convergence (in the case of 0, simulation will persist until the specific iterations)
- Relaxation Factor



1-2 Setting the iconCFD

Set the solver for the continuous phase at Solver Parameters panel.

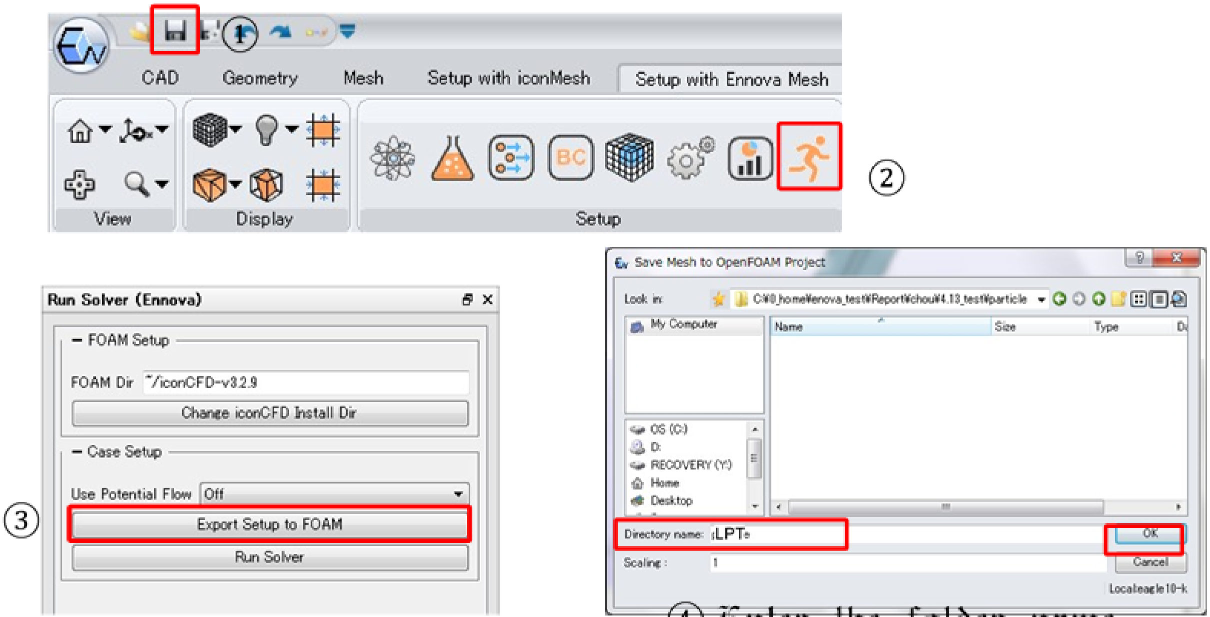


- Scheme
  - Differencing scheme
- Parallel Processing
  - Parallel settings for iconCFD

1-2 Setting the iconCFD

Saving the files for iconCFD solver

- Save the files set in ennovaCFD



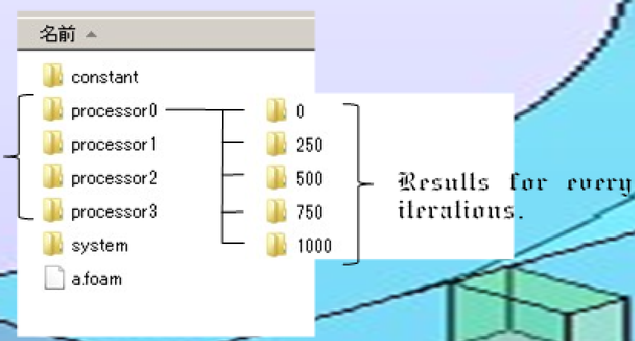
④ Enter the folder name

1-3 Running the

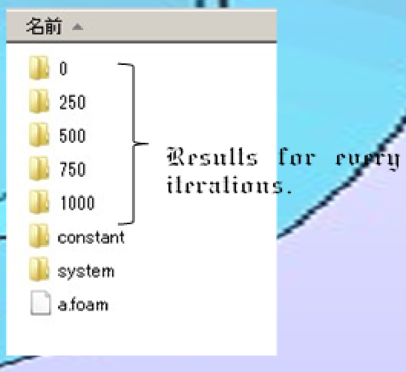
After the mesh and solver settings in the Windows terminal, copy the files to a system where iconCFD could run (such as simulation server).
Execute runHPC.sh to start simulations
After the simulations, copy the results to the original terminal (windows).

The folders of the following kinds will be generated after the simulations are finished. The output frequency is as specified in the Solver Parameter panel.

- In the case of multiple processors



- In the case of single processor



Results for each

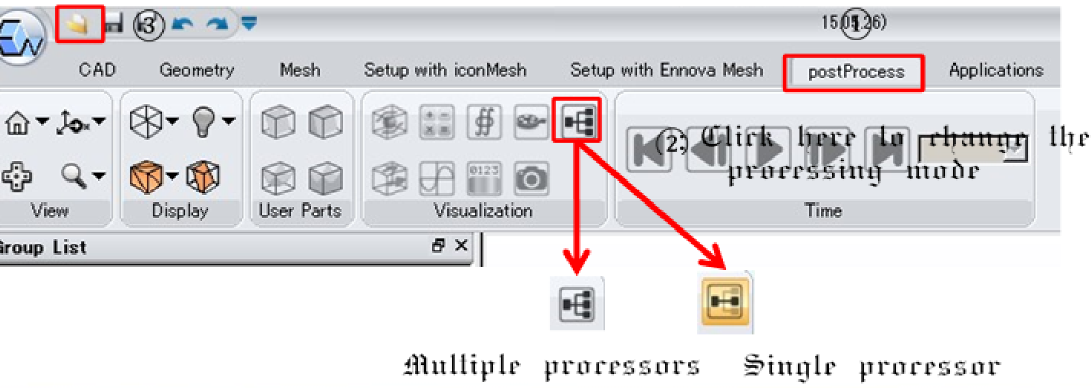
1-4 PostProcessing

Reading the simulation data from iconCFD（1/3）

- In the case of multiple processors
  - Put ***system*** ***constant*** ***processor**** ***.foam** files to a location accessible by Windows-PC.
- In the case of single processor
  - Put ***system*** ***constant*** ***time directories*** ***.foam files** to a location accessible by Windows-PC.

Selecting the postProcessing method

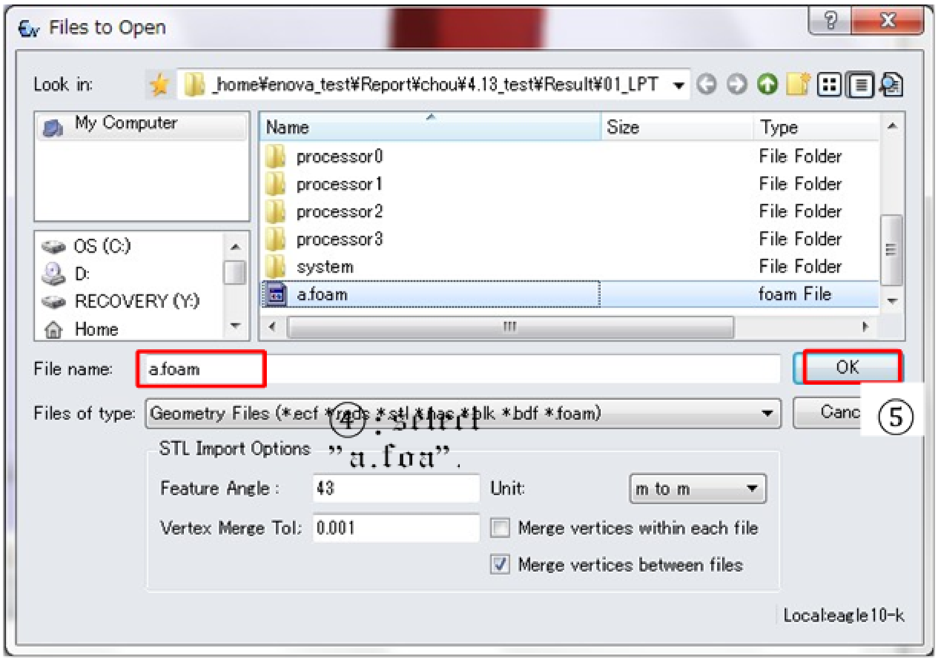
- Before reading the files, choose to process the results using multiple processors or single processor.



1-4 PostProcessing

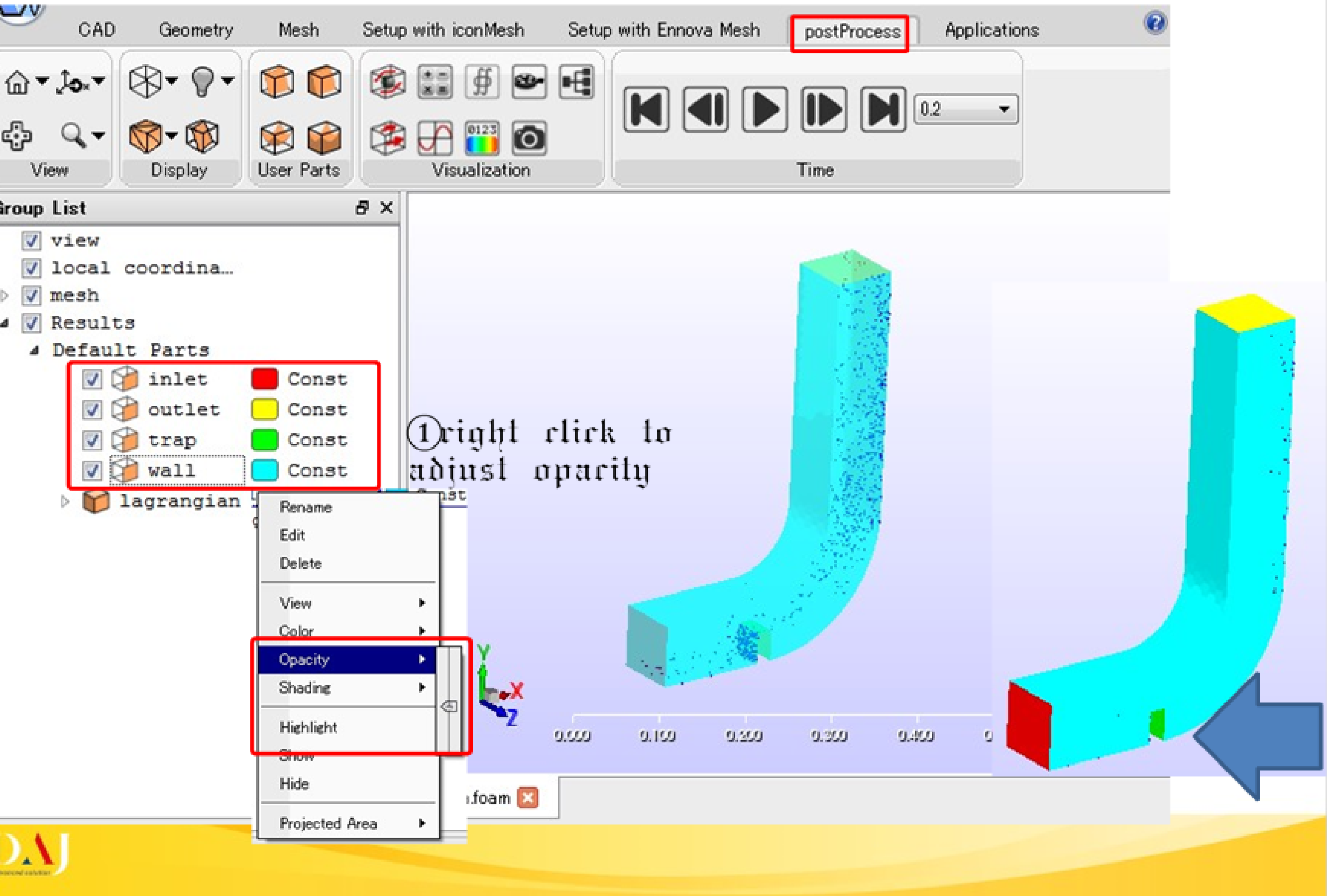
Read the output data of iconCFD 2/2)

- "a.foam" is automatically generated by runHPC.sh



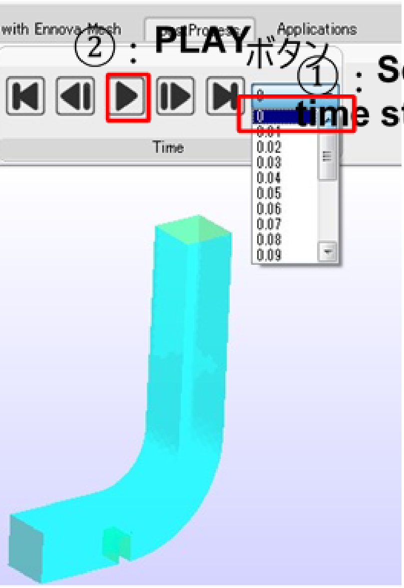
1-4 Postprocessing

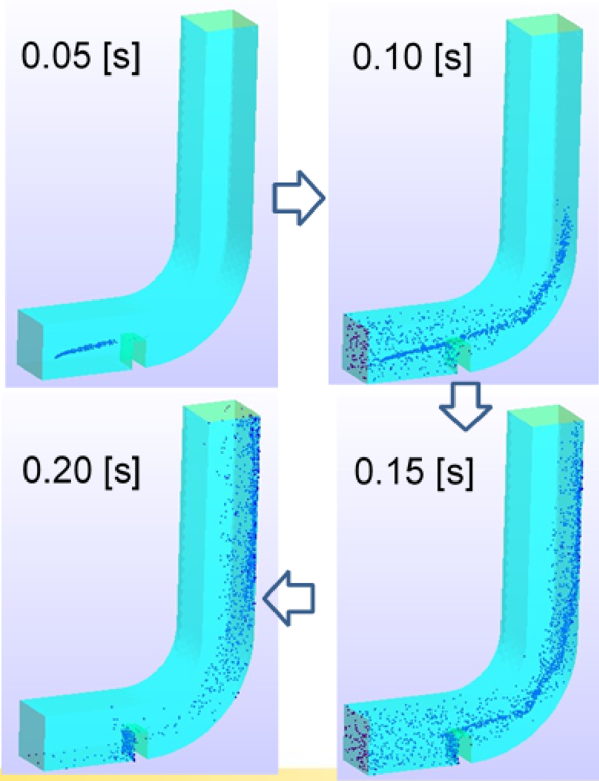
The particles could be visualized when the opacity of the boundary is changed.



1-4 Postprocessing

The animation of particles by reading the time history files in ennovaCFD.

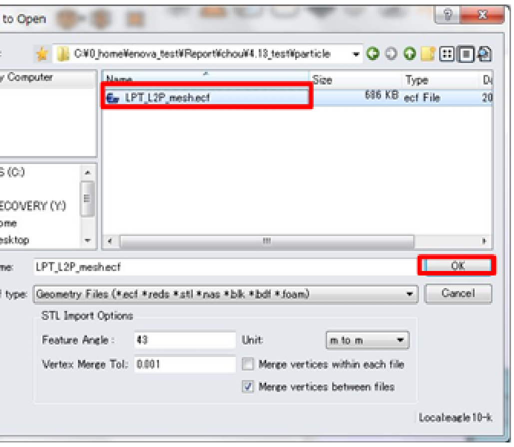


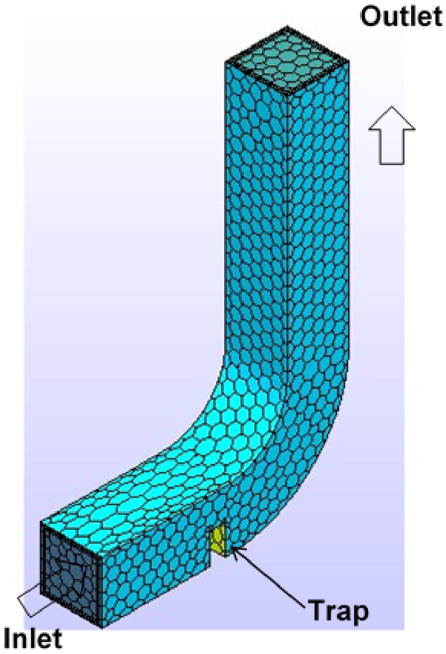


Settings for unsteady, coupled fluid-solid
rticle simulation

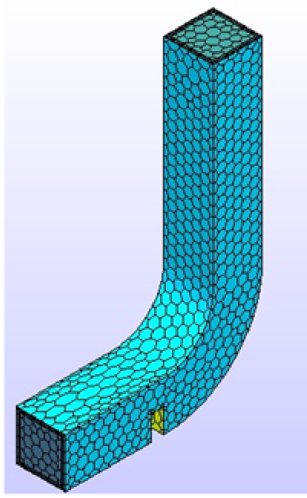
2-1. Importing the

Use the sample file LPT_L2P.ecf. The polyhedral mesh has already been generated.
Import LPT_L2P.ecf.



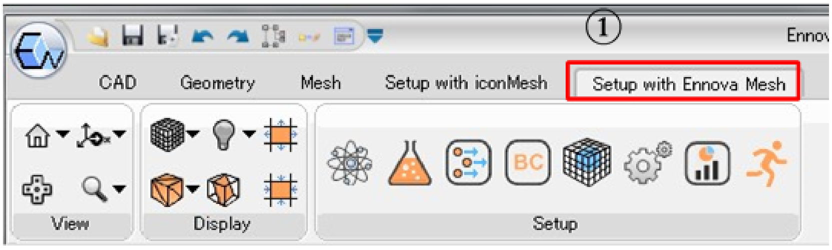


Polyhedral mesh



2-2 Setting the iconCFD Solver

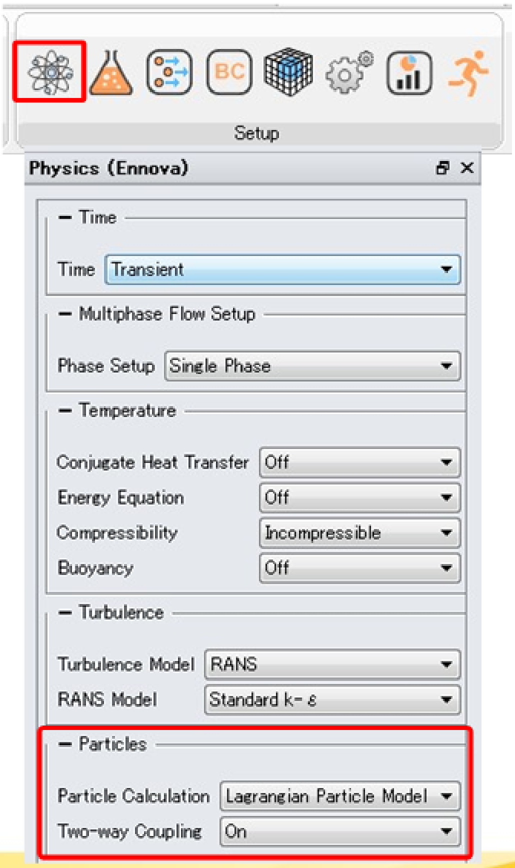
The solver settings will be in the Setup with Ennova Mesh panel.



2-2 Setting up the CFD Solver

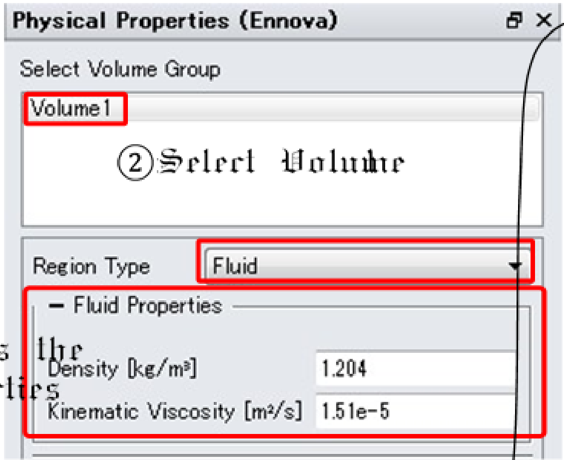
In the current case, the unsteady and fluid-solid coupled simulations will be carried out. In the Physics panel, the settings for the continuous phase (steady/unsteady, compressibe/incompressible, temperature, turbulence models) will be carried out.
The settings are as follows:

- Time: Transient
- Conjugate Heat Transfer: Off
- Energy Equation: Off
- Compressibility: Incompressible
- Buoyancy: Off
- Turbulence Model: RANS
- RANS Model: Standard k-ε
- Particle Calculation: Lagrangian Particle Model
- Two-Way Coupling: On



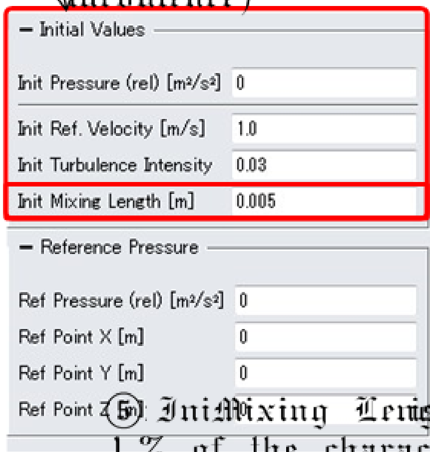
2-2 Setting the on CFD Solver

In the Physical Properties panel, the physical properties of the continuous phase will be specified. In the current case it is treated as air.



the Fluid
d, the
l value is the
al properties
ir

④ Initial conditions (pressu
turbulence)



⑤ Init Mixing Length set a
1 % of the characteristic
of the

2-2 Setting the CFD Solver

he settings for the particles in the Particles Settings panel.

- Injection tab

①

② Set the particle injection conditions for the Injection

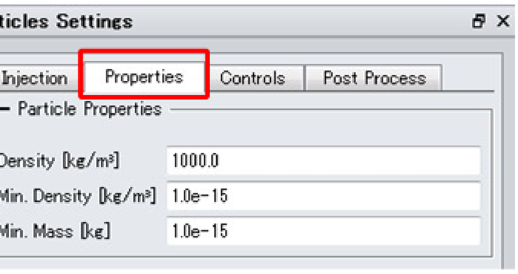
③ When injecting the particle from the boun
select Patch Injection the Detailed information
will be set at the Boundary Conditions

④ In the case of Cone shape spray
the parameters are set as

2-2 Setting theonCFDolver

he settings for the particles in the Particles Settings panel.

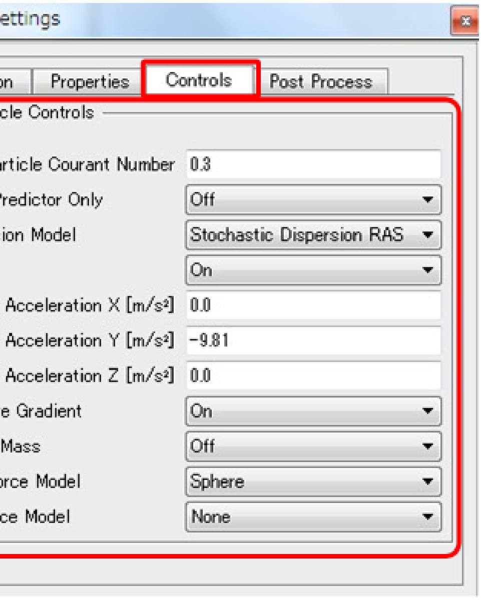
- Properties tab



2-2 Setting thieonCFDolver

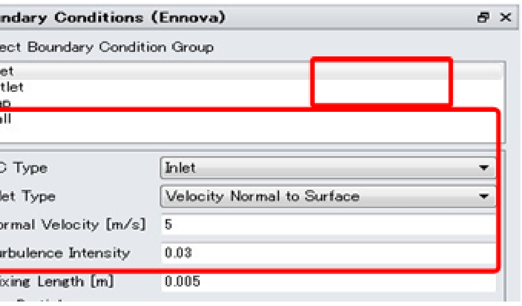
he settings for the particles in the Particles Settings panel.

- Explanation for the Controls tab



2-2 Setting thieonCFDolver

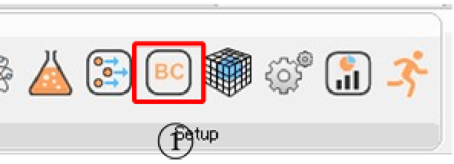
he settings for the particles in the Particles Settings panel.

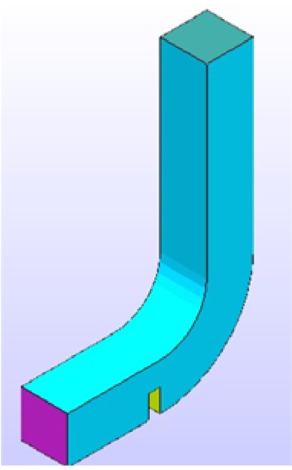


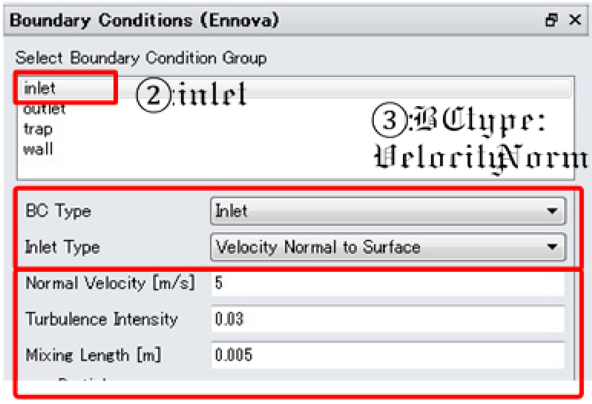
2-2 Setting thieonCFDolver

Boundary Conditions

- Set the boundary condition for the inlet.





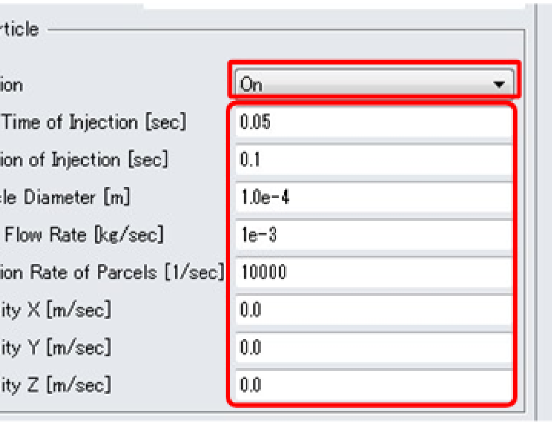


④:set the velocity and the turbulence intensity

2-2 Setting theonCFD Solver

Boundary Conditions

- Next, set the Patch Injection for the particles at the inlet.

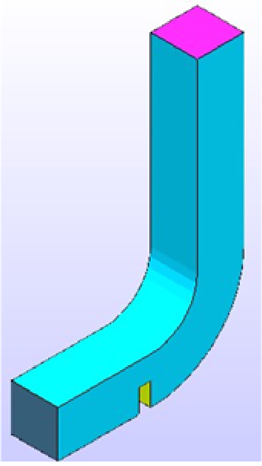


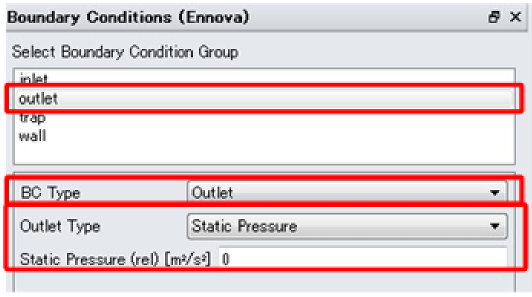
①Injection

2-2 Setting thieonCFDSolver

Boundary Conditions

- Set the outlet boundary condition for the continuous phase.





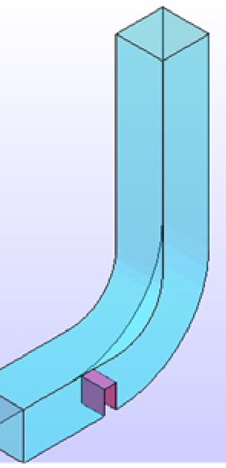
①:outlet

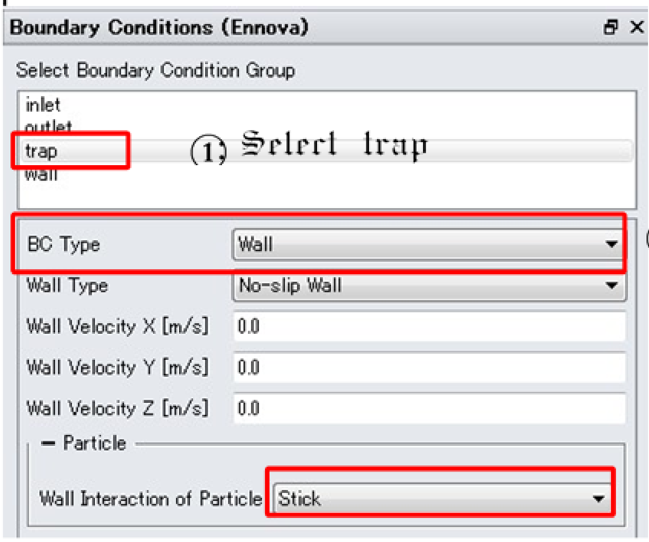
②:BCType: outlet

2-2 Setting the iconCFD Solver

Boundary Conditions

- Set the boundary condition of the continuous phase and the particle for the Trap.





① Select trap

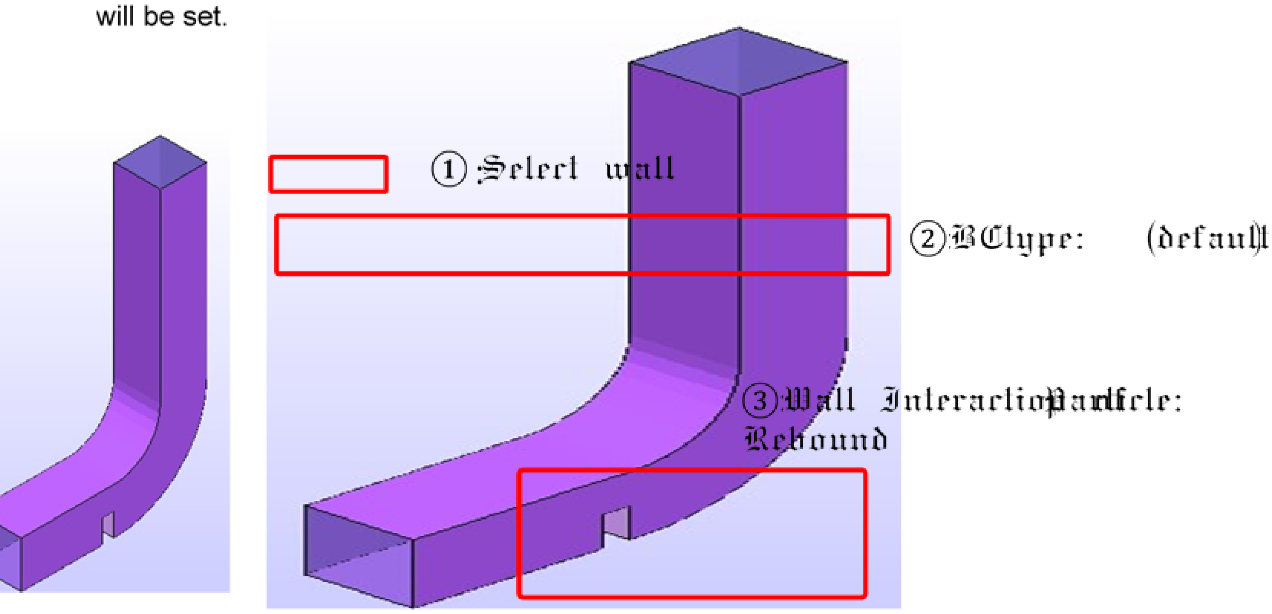
② Select Wall at BC type (Default)

③ Select Stick at Wall Interaction of Particle

2-2 Setting the CFD Solver

In the Boundary Conditions panel, the boundary conditions for every groups will be set.

- On the wall, the boundary condition for the continuous phase and the particle boundary condition will be set.



vp= (1-friction coeff)vpt – (restitution coeff)vpn

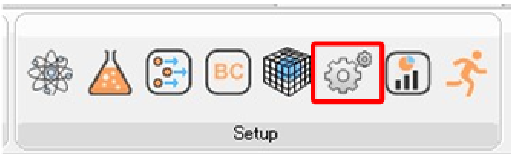
Vp: particle velocity vector after rebounding

Vpn: The normal velocity vector of the particle before rebounding

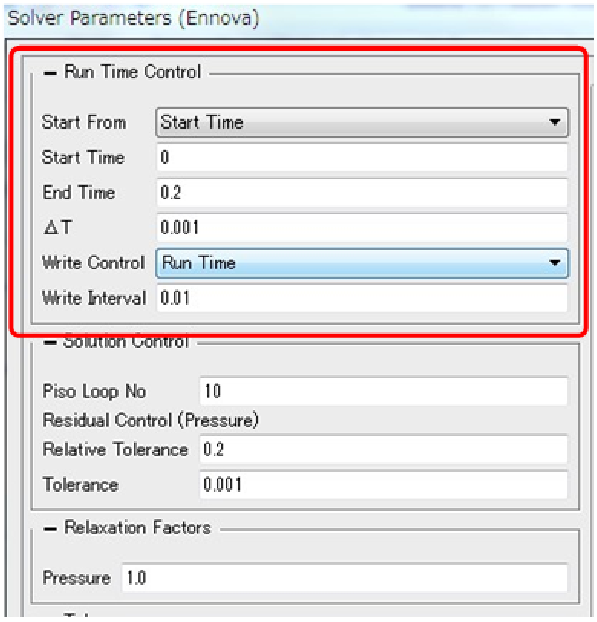
Vpt: The tangential velocity vector of the particle before rebonding

2-2 Setting the CFD Solver

Set the solver for the continuous phase at Solver Parameters panel.

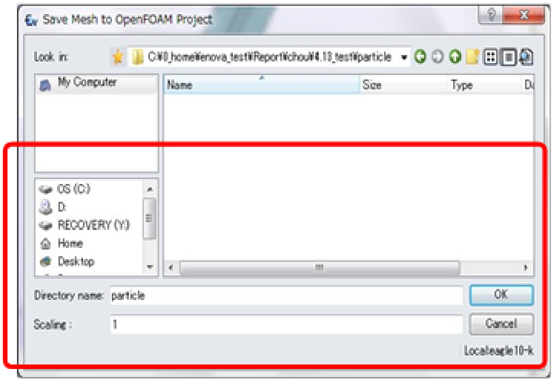


- Start Time/End Time
- ΔT
  - Time interval
- Write Control
- Write Interval
- Solution Control
  - PISO loops, tolerance
- Relaxation Factor



1-2 Setting the iconCFD

Set the solver for the continuous phase at Solver Parameters panel

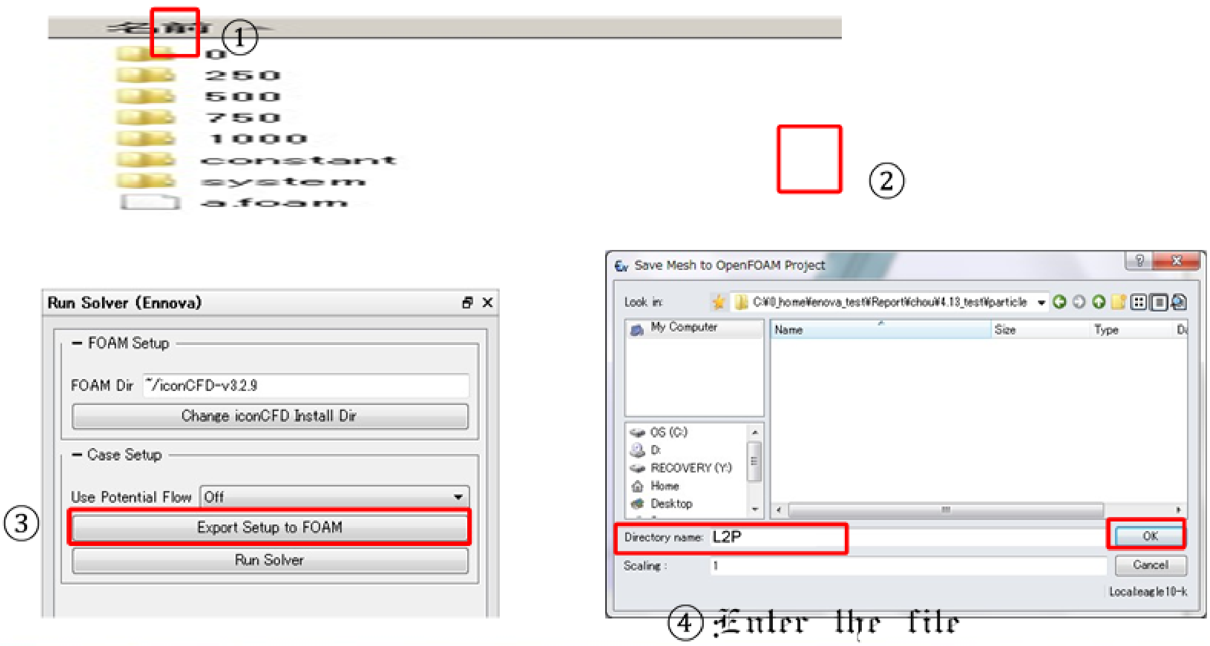


- Scheme
  - Differencing scheme
- Parallel Processing
  - Parallel settings for iconCFD

2-2 Setting the iconCFD solver

Save the files for the iconCFD solver.

.



2-3 Running the simulatipostProcessing

The following procedures are the same as in the previous chapter.

A Manual for Heat Transfer Analysis of conjugate heat transfer Using ennovaCFD v1.5

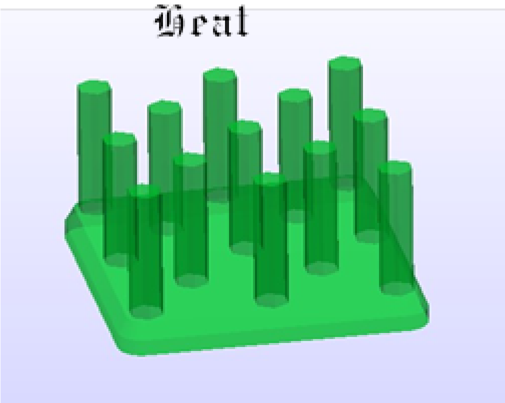
IDAJ Co.LTD

Contents

Introduction

This manual introduces the use of ennovaCFD v1.5 and iconCFD3.2.11 to carry out the heat transfer analysis of fluid-solid systems.

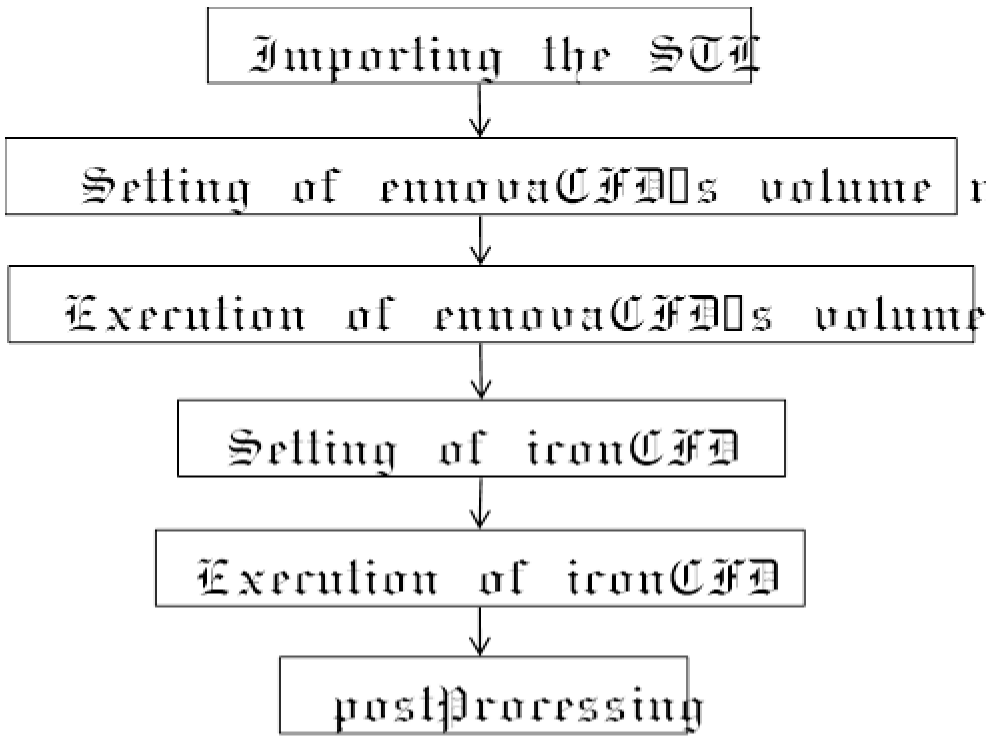
- The mesh generation is performed by the ennovaCFD's volume mesher (topologyBasedMesh).
- The steady-state natural convection analysis around a heat sink will be used as an example.



- Simulation overview
- Heat transfer analysis of fluid-solid system
- Steady-state
- Compressibility (ideal gas)
- Turbulent flow (standard k-ε model)
- Buoyancy force
- Heat source

1. Introduction (Contd)

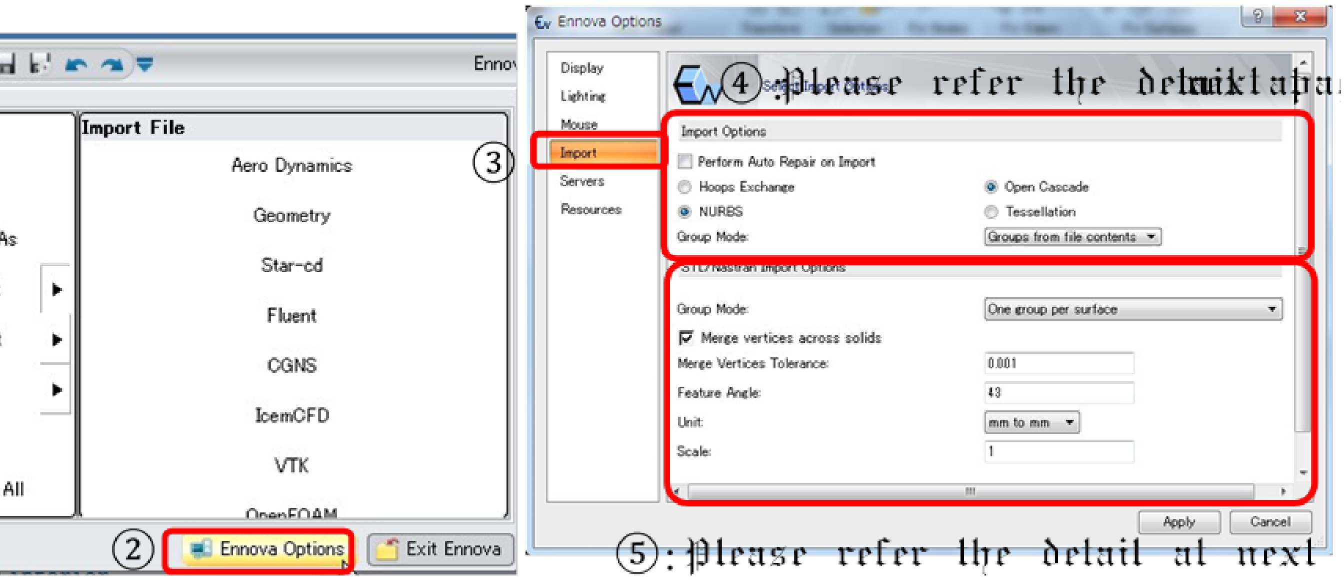
The procedure is as follows.



2. Importing the STL data

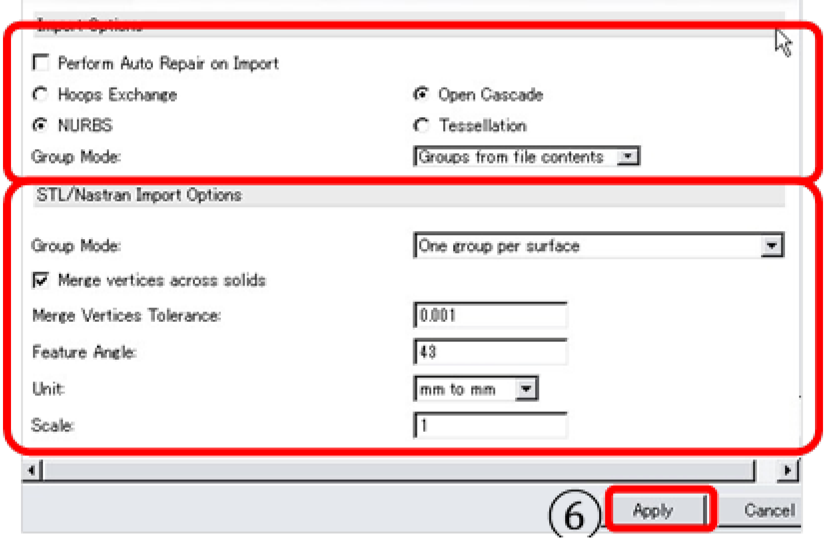
Setting of the import options (1/5)

1. To open the import options, click the ennova icon .
2. Click the ***Import*** tab to display the import options.



Setting of the import options (2/5)

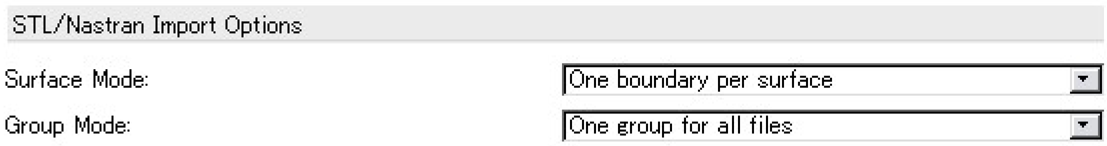
- Import Options
- Uncheck the ***Perform Auto Repair on Import***（for the β functionality）.
- Check the ***Open Cascade*** .



- Leave the other settings as default.
- STL/Nastran Import Option (Details will be given in the pages that follows).
- Group Mode　　One Group per surface
- Merge Verticies across solids　　On
- Merge Vertices Tolerance　0.001
- Feature Angle　43
- Unit　mm to mm
- Scale 1

Setting the import options (3/5)

- Explaination of ***STL/Nastran Import Option***



- Group Mode:
- One group per file
- When importing multiple STL files, create the geometry groups for every STL files.
- One group for all files
- When importing multiple STL files, create one geometry group for all the files.
- One group per surface
- Create geometry groups for every sub-part of the STL file.

Setting the import options (4/5)

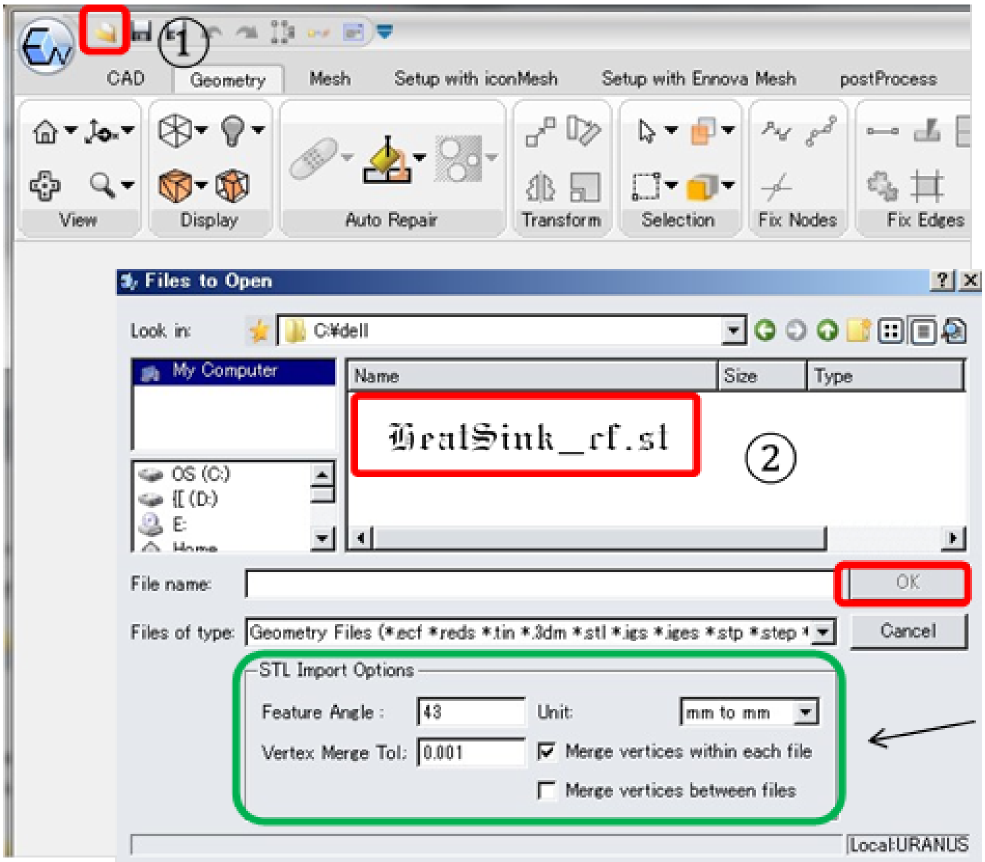
- Explaination of ***STL/Nastran Import Option***

- ***Merge vertices across solids***
- When importing multiple files, merge the vertices within the tolerance.
- This options is used when ***One group per file/One group*** for all files is selected.
- ***Merge vertices Tolerance.***
- Merges the gaps below this value.

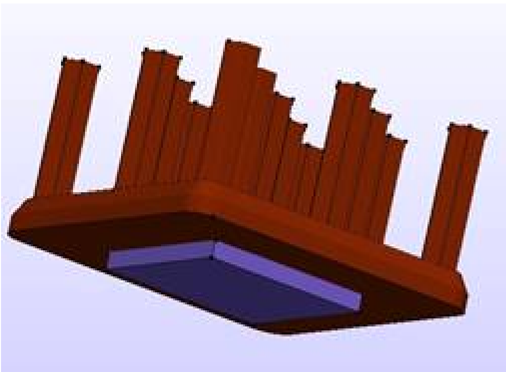
Setting the import options (5/5)

- Explaination of ***STL/Nastran Import Option***

- Feature Angle
- Set the minimum angle between polygonal facets when displaying feature lines.
- Unit
- Scale the model’s size by unit.
- Scaling with the same unit (e.g., m to m) does not change the size of the model.
- Scale
- Scale the model’s size with specified ratio.
- No scale if specified to 1.



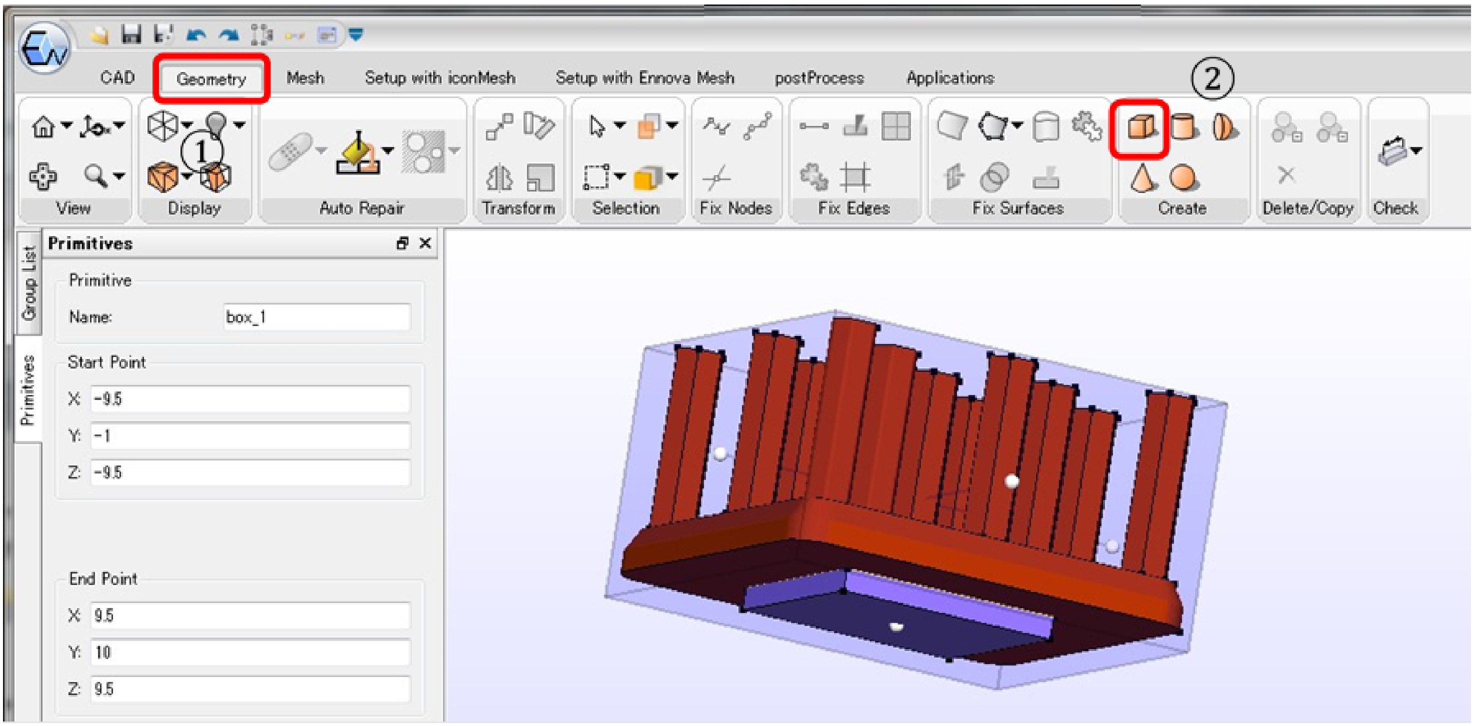
Here shows the o
we have set in
slides.



Import the STL file (HeatSink_cf.stl).

3. Creating the computational domain

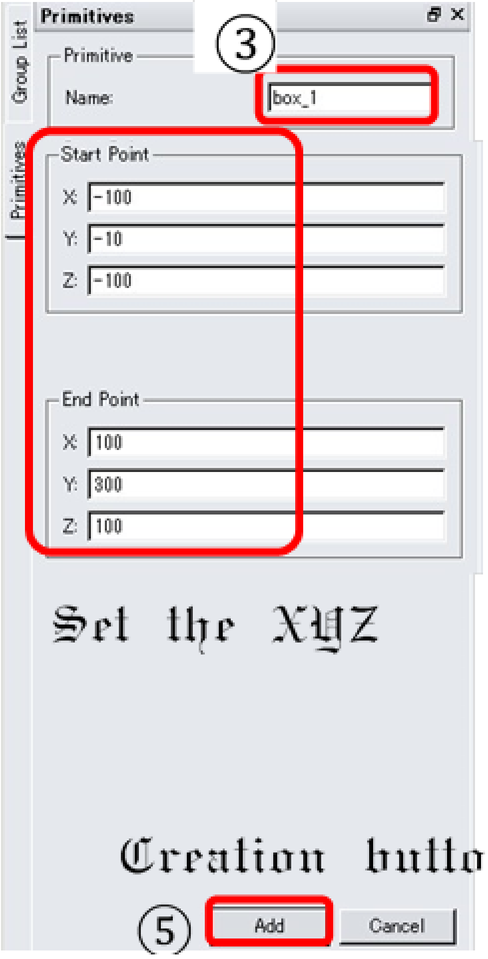
Creating the outer boundaries using the primitive functionality.

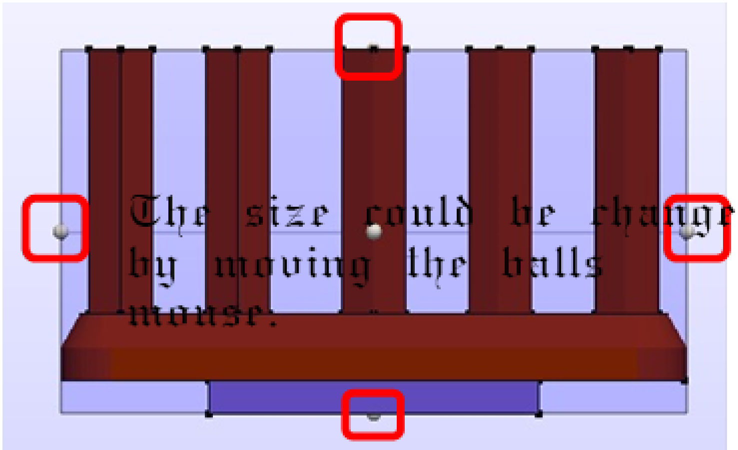

CAD
Geometry
Mesh
Setup with iconMesh
Setup with Ennova Mesh
postProcess
Applications
①
②
View
Display
Auto Repair
Transform
Selection
Fix Nodes
Fix Edges
Fix Surfaces
Create
Delete/Copy
Check
Group List
Primitives
Primitive
Name: box_1
Start Point
X: -9.5
Y: -1
Z: -9.5
End Point
X: 9.5
Y: 10
Z: 9.5

##Creating the computational domain

Input the size of the box, or move the balls on the GUI.

Set the names for the geometry groups.





This manual uses the size specified by ④ to create the outer boundaries.

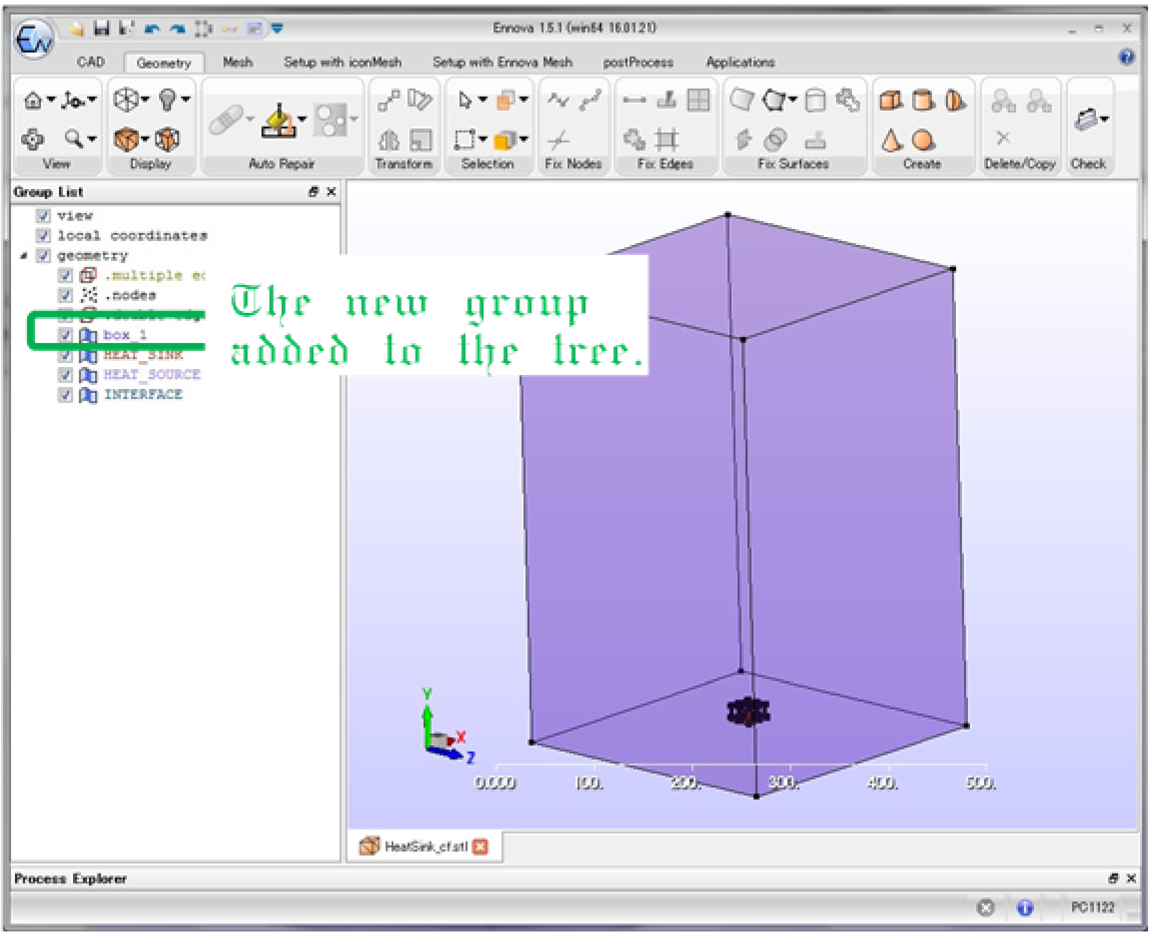
④

##Creating the computational Groups

Confirm the created computational domain

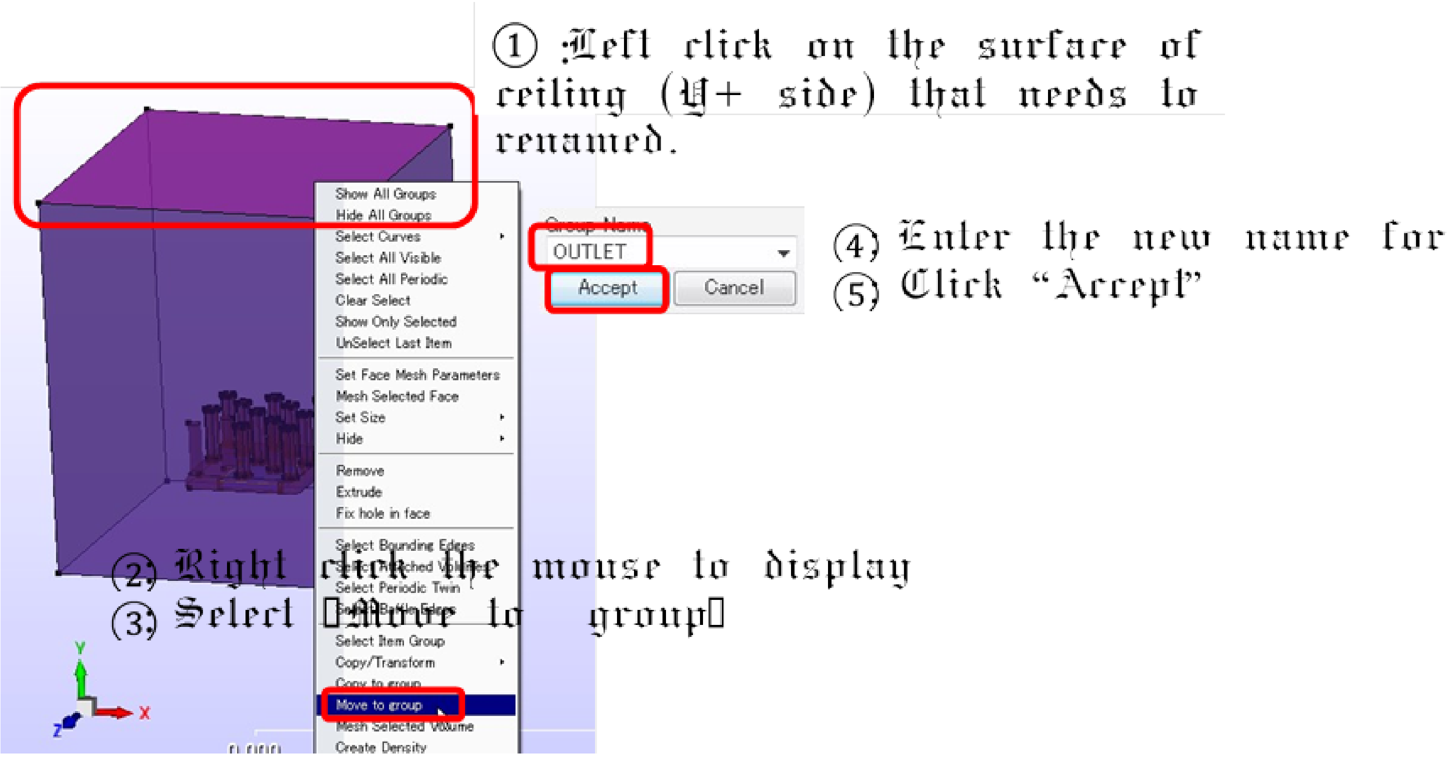
n A new geometry group is added to the group list tree when the outer domain is

created.



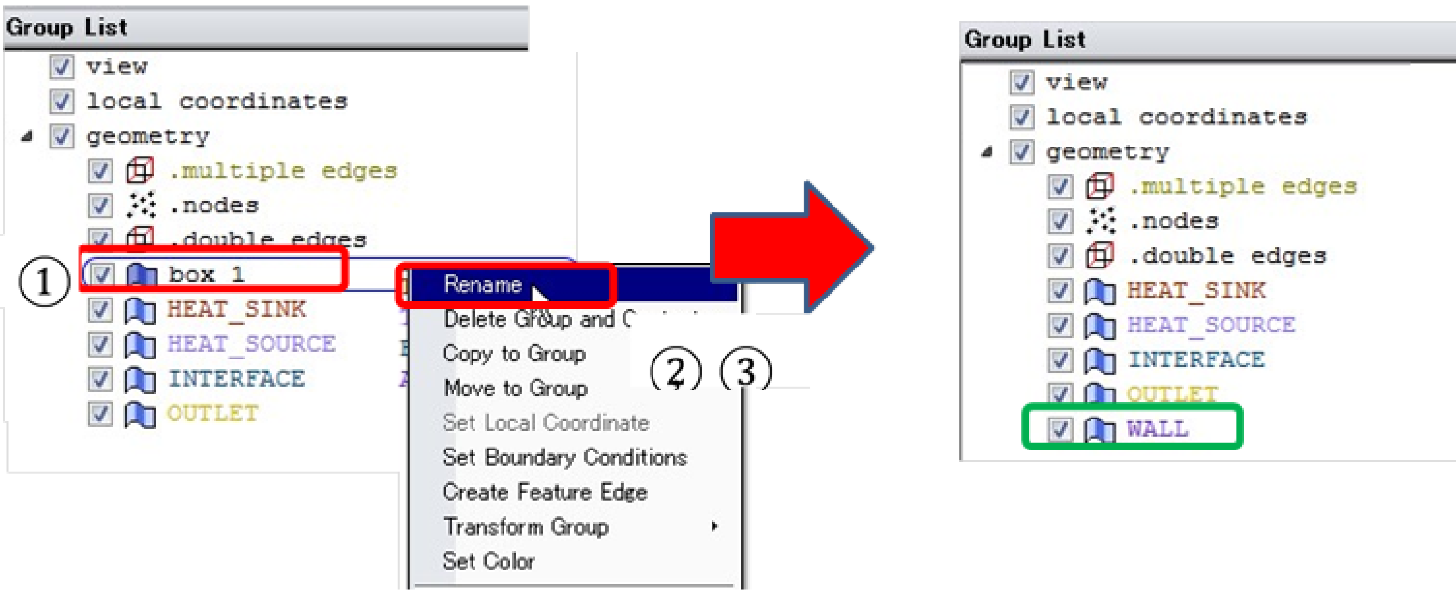
4. Changing the names of the group

Change the group name for the ceiling of the outer domain.



4. Changing the names of the group (Contd)

Rename the remaining part of box_1 to WALL.

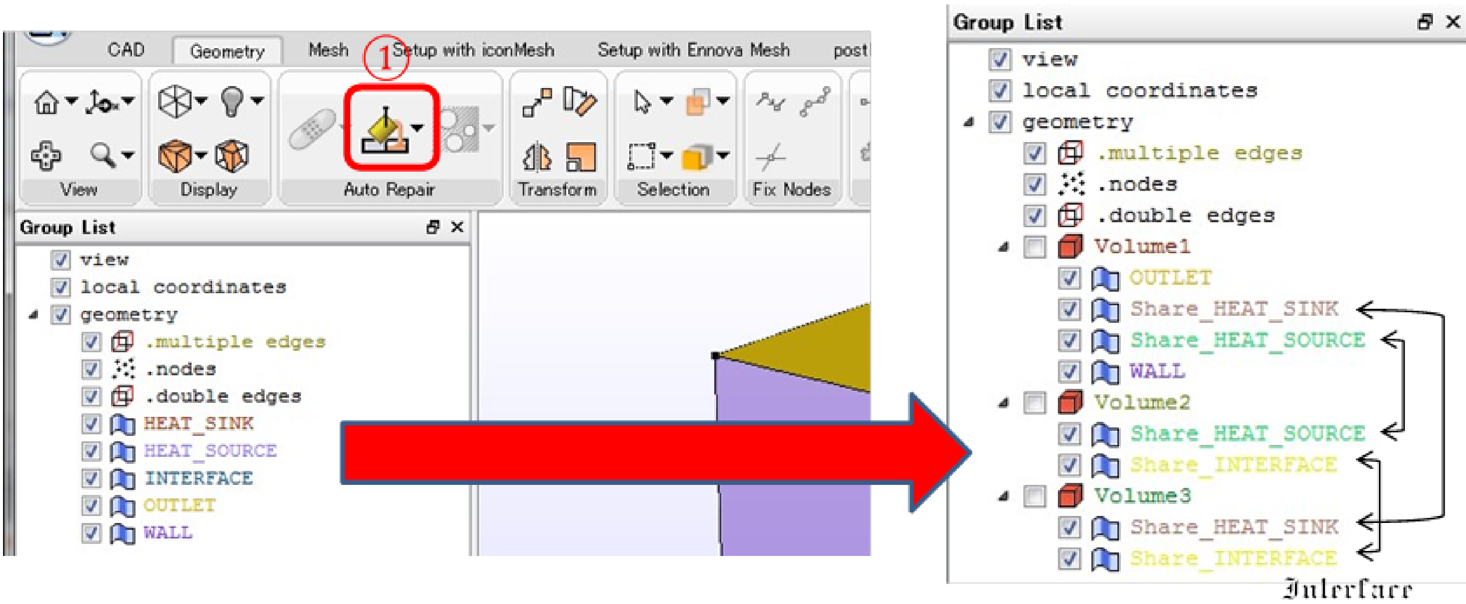


①: Select the item that needs to be renamed from the tree
②: Right click mouse to display menu
③: Click “Rename”
④: Enter the new name

##Creating the Volumes and Mesh

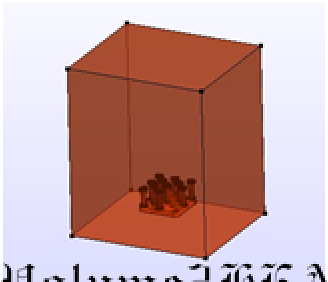
About the computational domain

- In ennovaCFD, we call the computational domain Volume.
- It is possible to use the FindVolume functionality to automatically find the volumes (see P17-19 for detailed information).
- In the current case, 4 volumes are automatically recognized.
- On the connection of 2 volumes, 2 surfaces with the same name are created.
- In ennovaCFD, we call these surfaces interfaces.



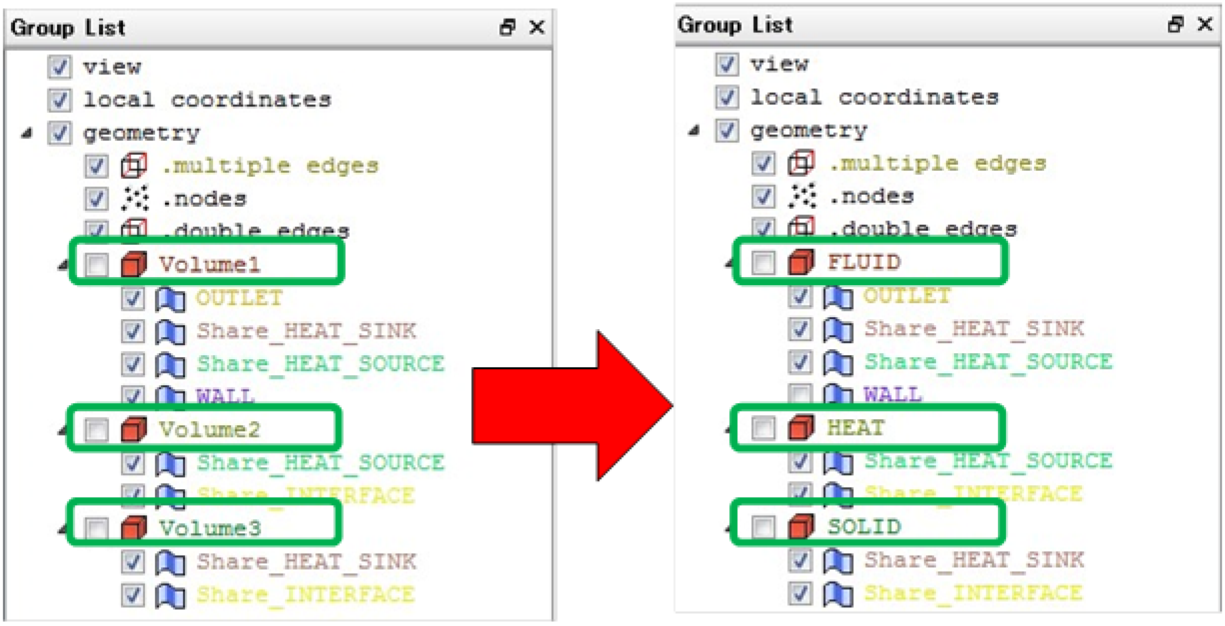
Rename the volumes as follows.

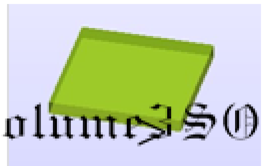
- Volume1 → FLUID   Volume1＞FLUID
- Volume2 → HEAT (heat source)



n Volume2 HEA

display the menu, and select “Rename”.





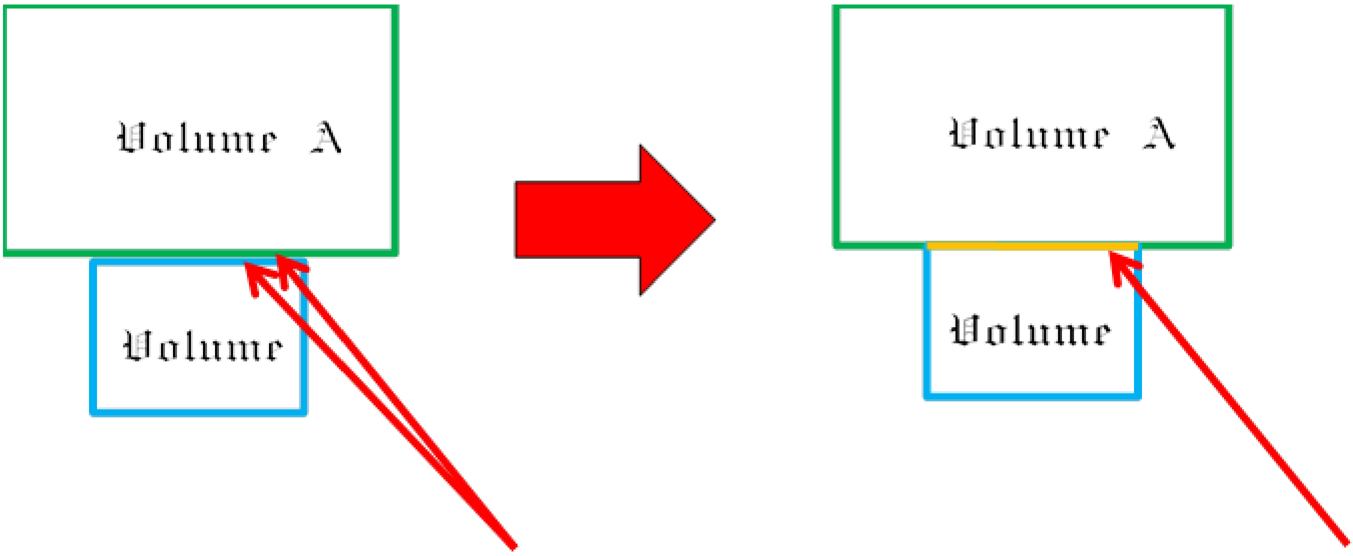
Volume3 SOL

Volume3 → SOLID

n Right click on the volumes that need renaming to

Requirement of the STL file for automatic volume detection Volume interfaces need to be integrated to be shared surfaces.

CAD data STL file

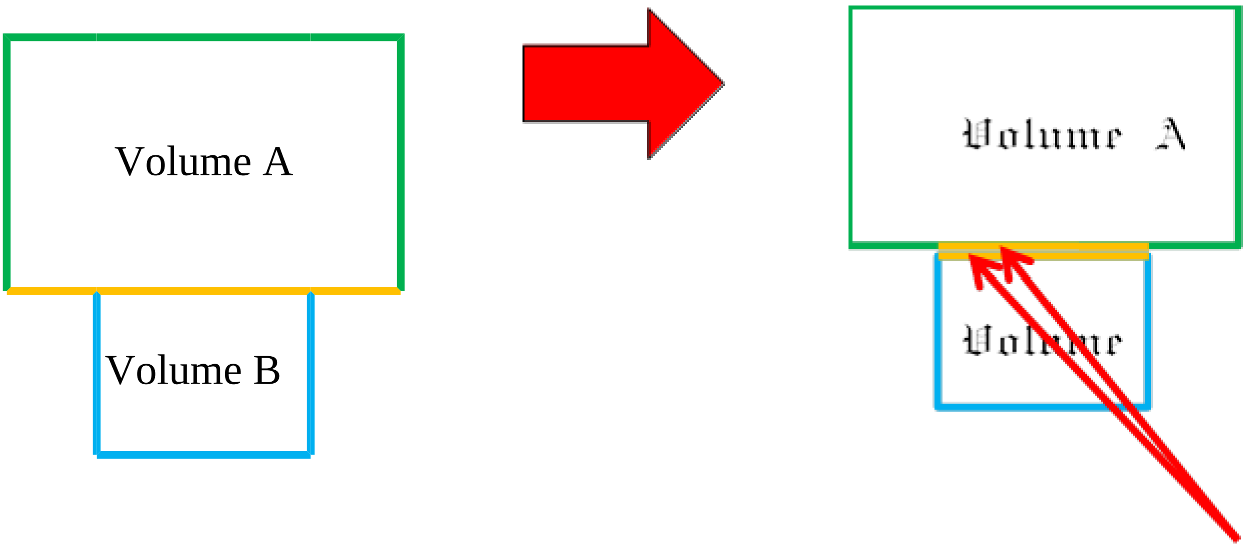


Each Volume owns their The two volumes share own surfaces. one surface.

Requirements of the STL file for automatic volume detection

- When executing FindVolume, the shared interface between the two volumes is automatically copied to allocate to the two volumes.

STL file The outcome of automatic detection

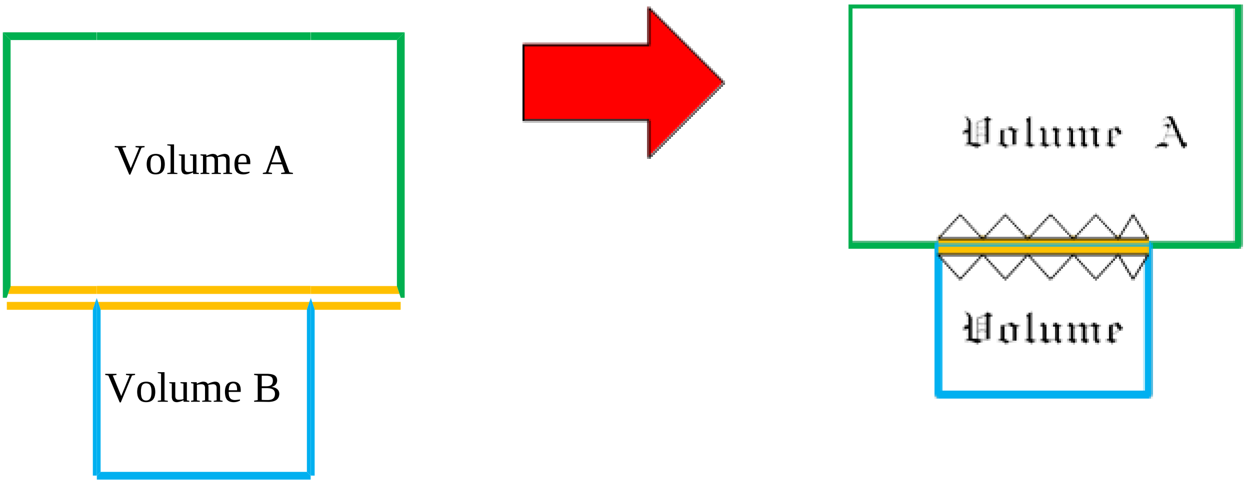


One shared interface is copied to make two surfaces.

Requirements of the STL file for automatic volume detection

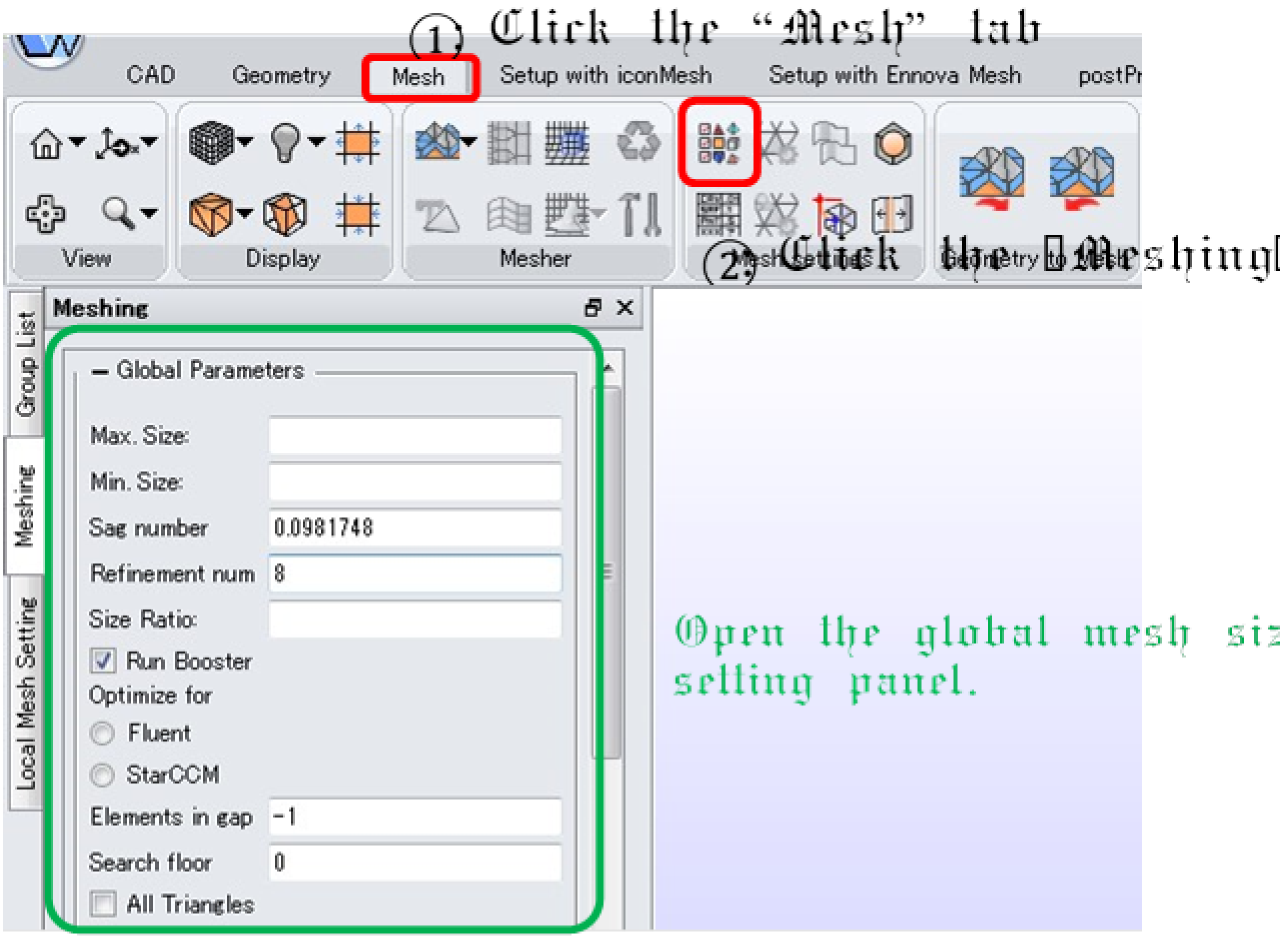
- On the interfaces created by FindVolume, continuous mesh is created across the shared surface.

The outcome of automatic detection Generated mesh



6. Setting of the mesh size

- Setting of the global mesh size

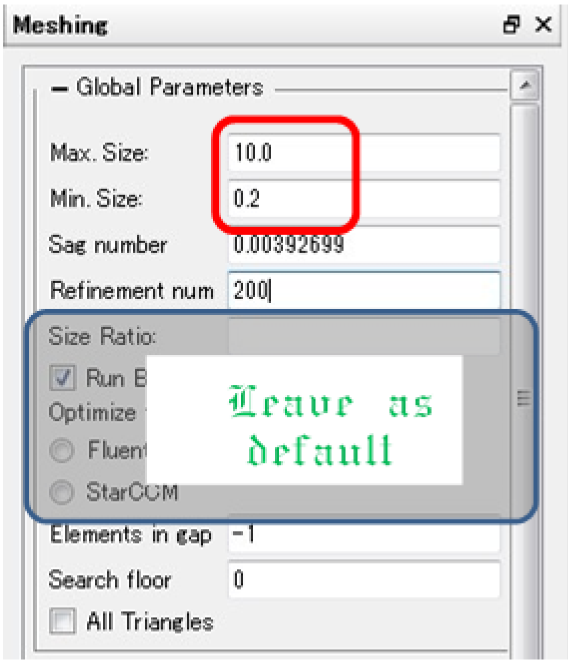


Details of the global mesh setting panel (1/3)

- In the current case we set the values as are shown in the red box.

①: Set the Max.Size to 10

Maximum mesh size for the model



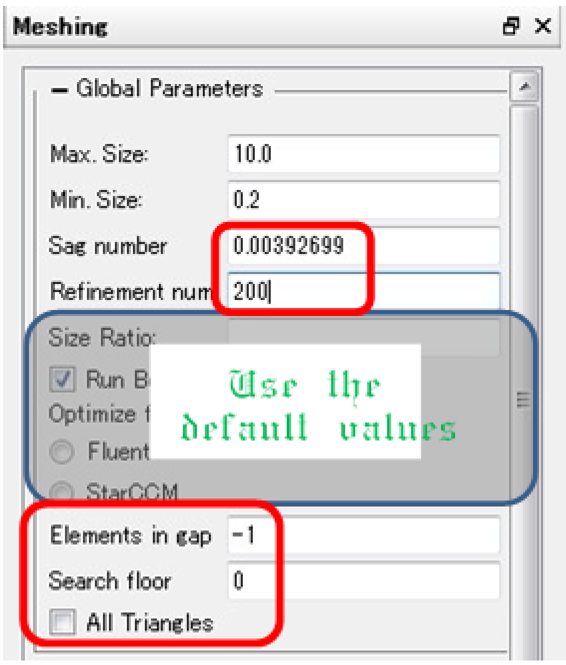
①②: Set the Min. Size to 0.2

② Mimimum mesh size for the model

Note: In ennovaCFD 1.5, the local mesh size can not exceed the maximum mesh size specified above. So, the maximum mesh size should be set based on the scenario that generate the largest meshes. Later, the mesh size could be refined using the local mesh setting.

Details of the global mesh setting panel (2/3)

- Set the values as are shown in the red box



①: Refinement num 200 fit ratio for the mesh on the curve surfaces See P25 for more details

②: Elements in gap -1 (default value)

① See P26 for more details

③: Search floor 0 (default) See P26 for more details

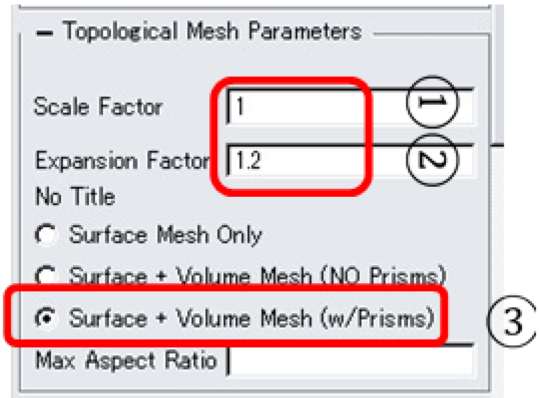
④: All Triangles off
If this option is turned on, the surface mesh will

②
all be triangles.

③
If this option is turned off, rectangular meshes will ④ be created where possible, the other places will be
meshed with triangles.

Details of the global mesh setting panel (3/3)

- Set the values as are shown in the red box



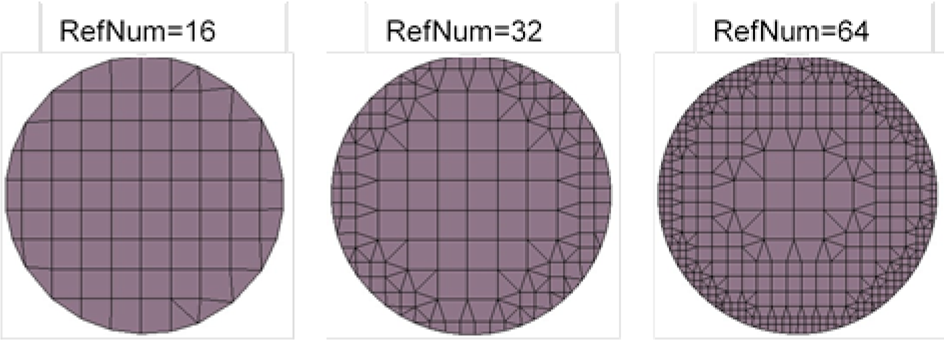
①: Scale Factor 1 (default) Scale the specified mesh size

②: Expansion Factor 1.2 (default)
Expanding ratio of the volume mesh

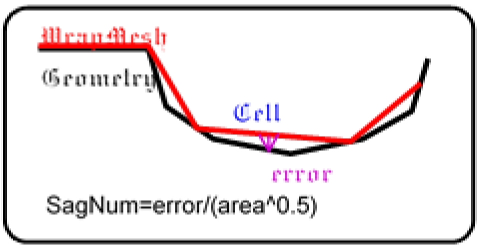
③: select Surface+Volume Mesh(w/Prisms) Selection of the mesh type

- Explanation of Refinement Number and Sag Number

Refinement Number
The estimated value of nodes on a circle when splitting. The larger the value, the finer the mesh.



Sag Number
Wrapping error divided by the squared root of the cell area. The lower the value, the finer the mesh

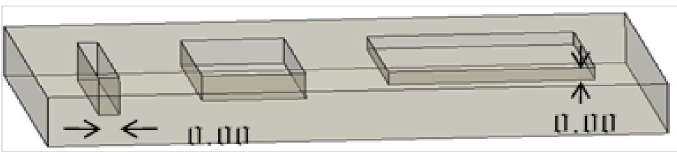


In addition, if one of the parameter is decided, the other one is also fixed.

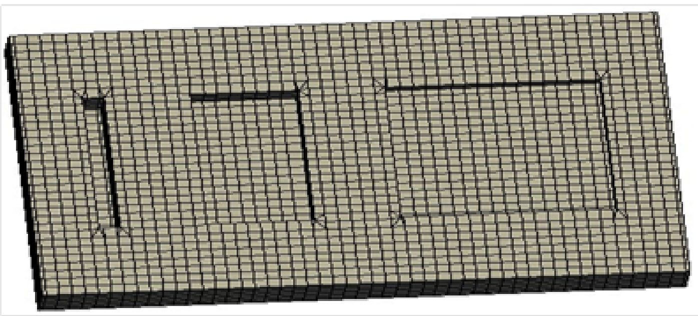
Refinement Number = π/Sag Number/4

If the **Min. Size** is left blank, the minimum mesh size will be decided by the scale of the model.
Refinement is restricted by the specified minimum mesh size.

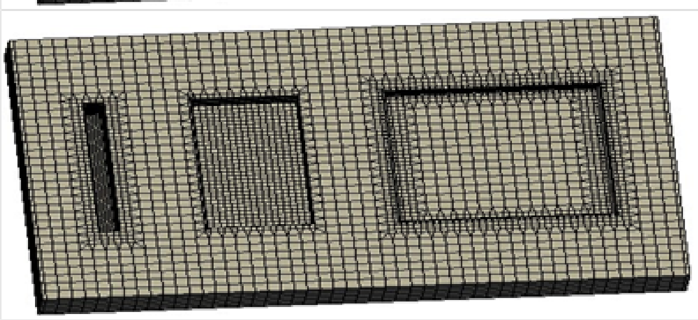
Explanation of Element in gap and Search floor



Example of
Max Size:
Min Size:



<default value (no >
Element in -1
Search floor:

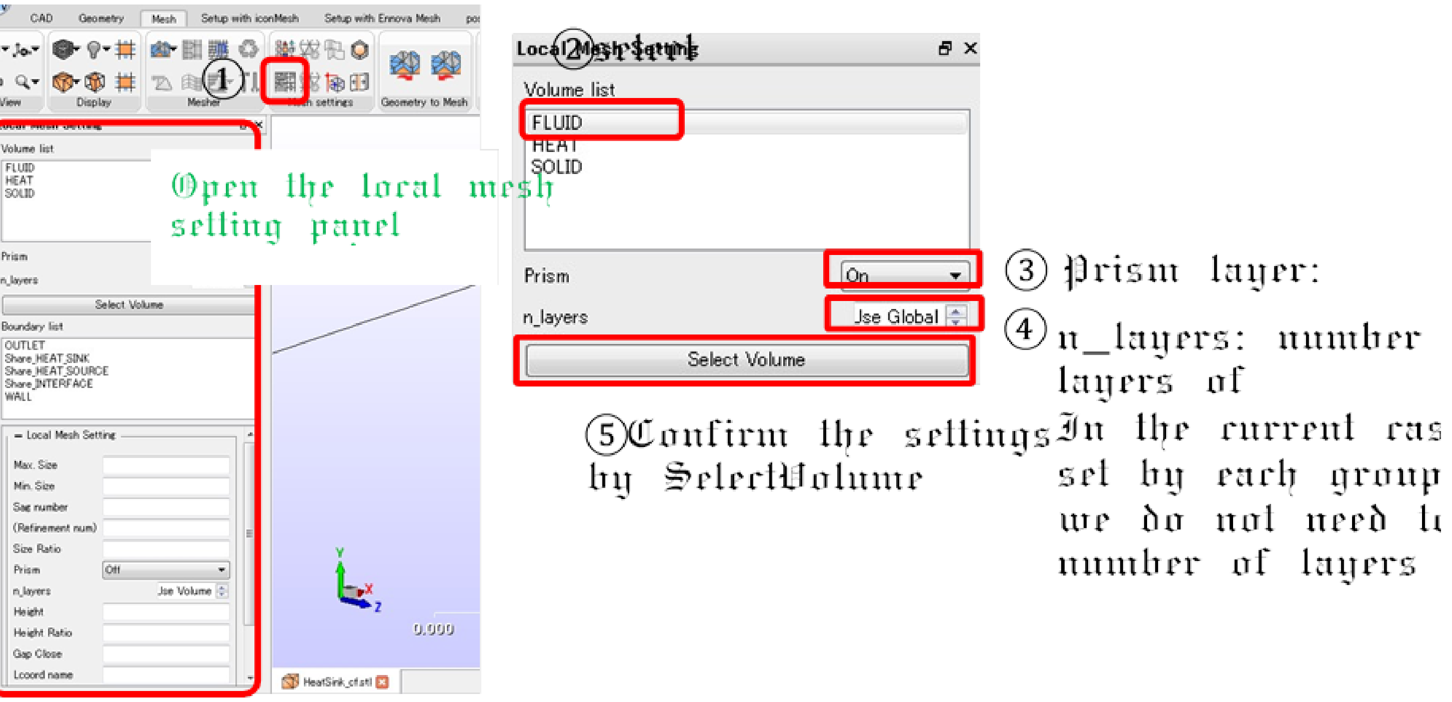


<4 subdivisions within a width
Element in gap:
Search floor:

When Min. Size is left blank, the smallest mesh size is: Search floor / Element in gap

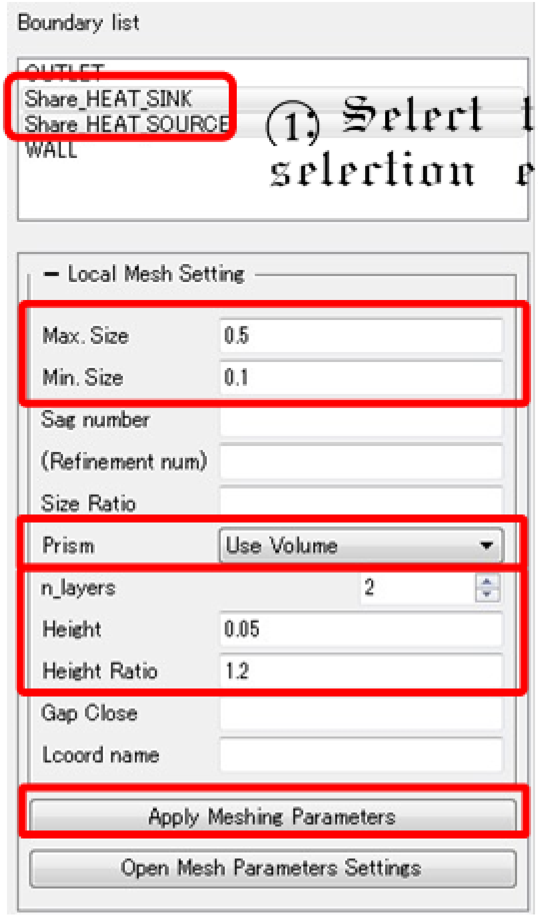
Settings of the local mesh size and layers. (1/3)

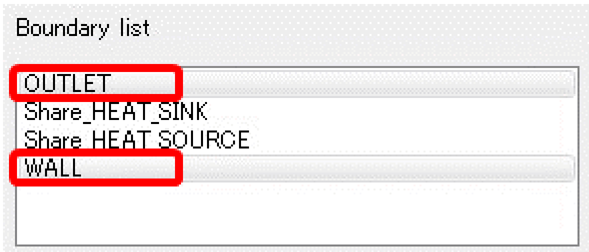
- On the solid surface in the fluid volume, we set the local mesh size and the layers.



Settings of the local mesh size and layers(2/3)

- In Share_HEAT_SINK and Share_HEAT_SOURCE, we set the value of the local mesh size and the layers.

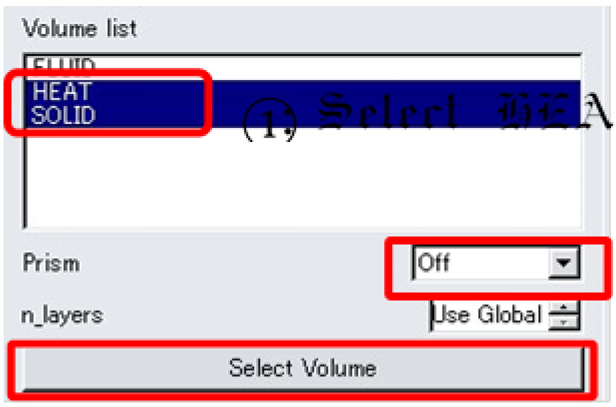




As for OUTLET and WALL, we do not perform local mesh settings.

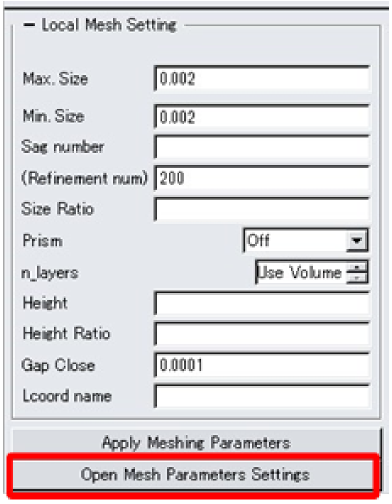
Settings of the local mesh size and layers (3/3)

- In the solid domain, turn the layer mesh off.



Confirmation of the local mesh settings and the layers

- It is possible to check the local mesh size settings by the display list.
- It is also possible to change the settings by directly modifying in the spreadsheet.



①

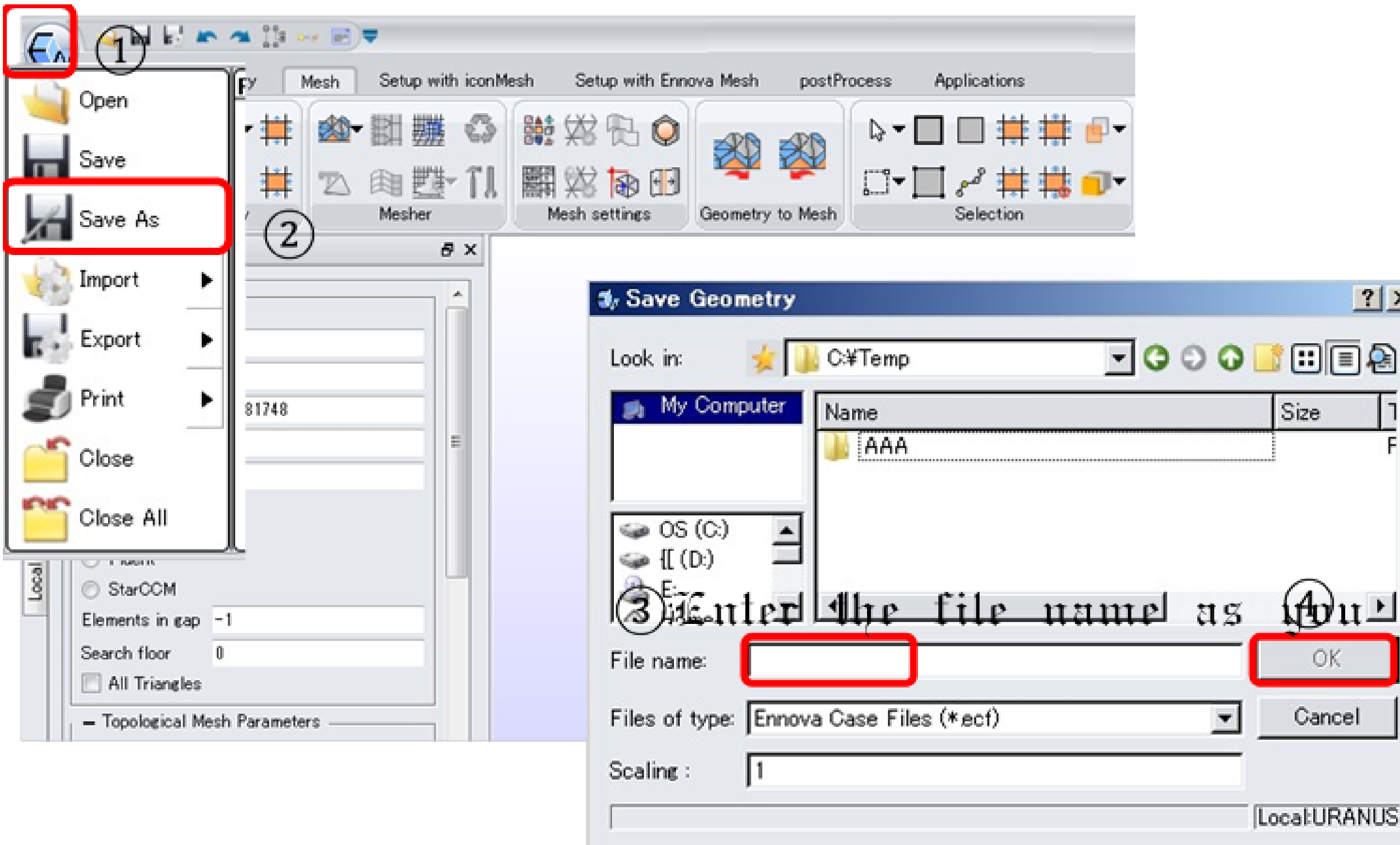
Mesh Setting Parameters

	active group	Max. Size	Min. Size	Sag number	Refinement num	Size Ratio	Prism	n_layers	Height	Height Ratio
al		10.0	0.1	0.0981748	200.0		☑	0		
ups										
FLUID							On	Use Global		
OUTLET							Off	Use Volume		
Share_HEAT_SINK		0.5	0.1				Use Volume	2	0.05	1.2
Share_HEAT_S···		0.5	0.1				Use Volume	2	0.05	1.2
WALL							Off	Use Volume		
HEAT							Off	Use Global		
Share_HEAT_S···		0.5	0.1				Use Volume	2	0.05	1.2
Share_INTERFA···							Off	Use Volume		
SOLID							Off	Use Global		
Share_HEAT_SINK		0.5	0.1				Use Volume	2	0.05	1.2
Share_INTERFA···							Off	Use Volume		

##Generating the Case

Save the case data

- Before mesh generation it is advised to save the data into .ecf file.
- The ecf file holds all the information for the ennovaCFD.



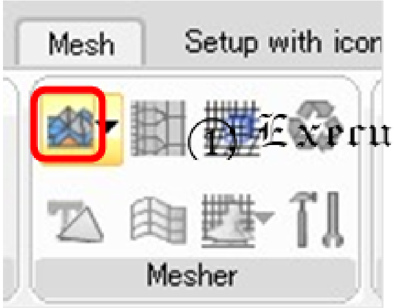
Volume mesher in ennovaCFD

- The Topology Based Mesh, which is based on the Delaunay method, is available in ennovaCFD for the volume mesh.
- Topology Based Mesh could be used to create models with multiple area, such as the fluid-solid heat transfer model.
- ↑ Creating the polyhedral mesh is also possible.

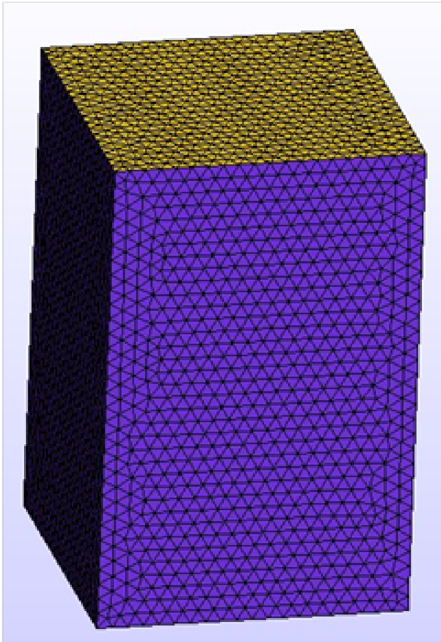
Button of Topology Based Mesh

Check it if polyhedral mesh is to be created（In the current case it should

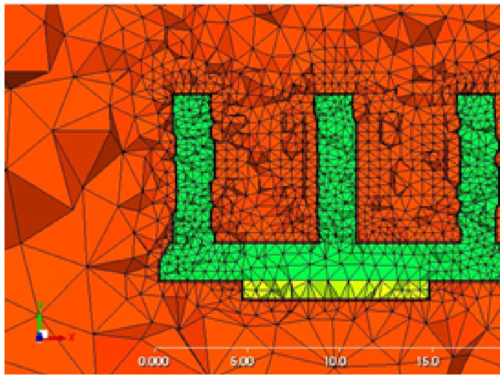
Mesh generation of domain with multiple areas



(1) Executing the volume mesher

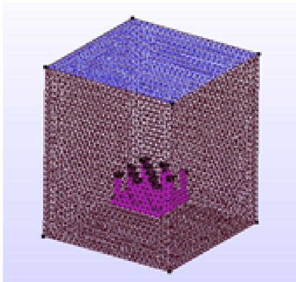


Generated mesh

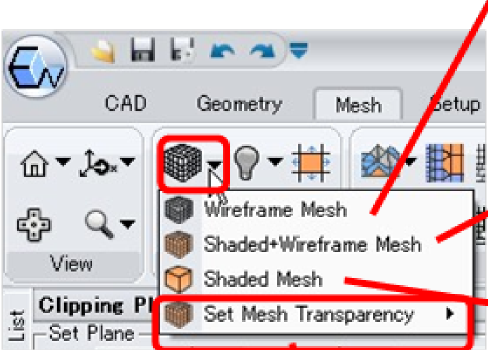


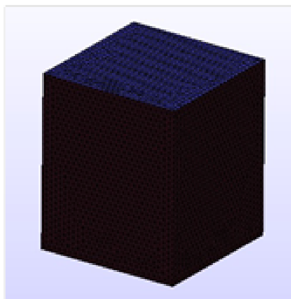
Change of displaying method

- Here we introduce the methods of
- checking the generated mesh.

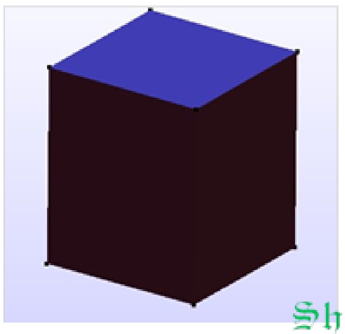


Wire frame

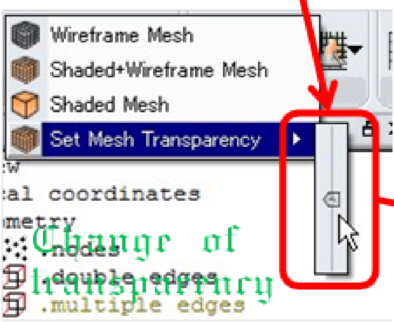




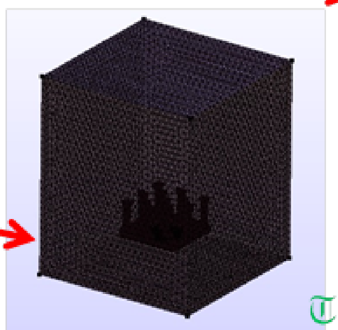
Wire + Sh
Default



Shade

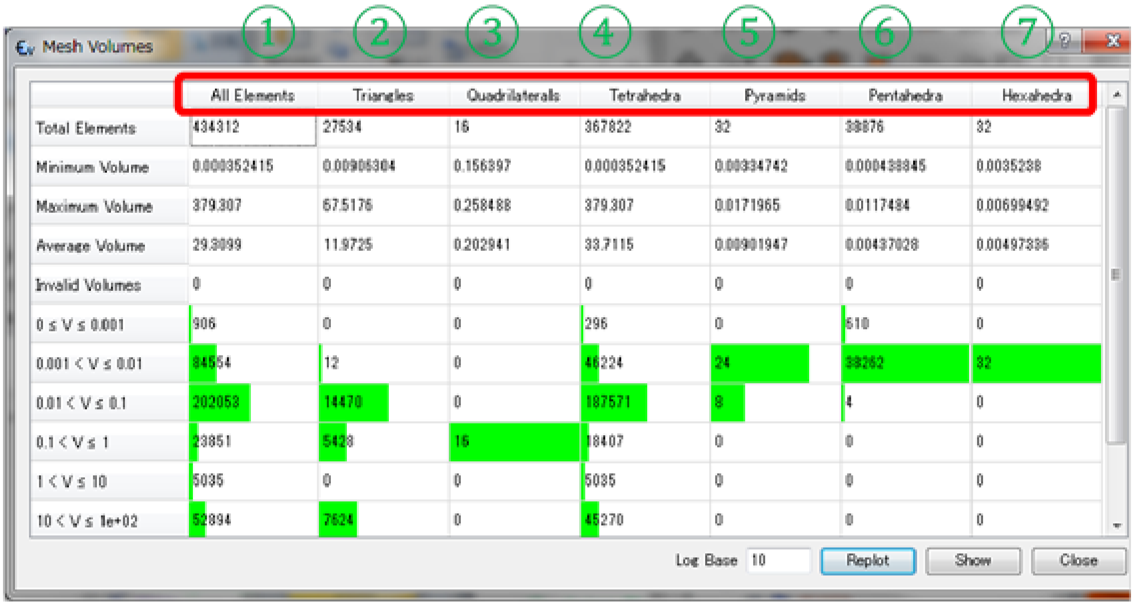


Change of transparency



Transparent

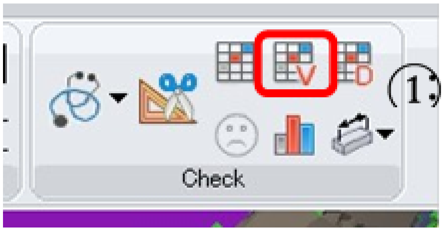
Check the mesh numbers



Mesh Volumes

	All Elements	Triangles	Quadrilaterals	Tetrahedra	Pyramids	Pentahedra	Hexahedra
Total Elements	434312	27534	16	367822	32	38876	32
Minimum Volume	0.000352415	0.00906304	0.156397	0.000352415	0.00334742	0.000438845	0.0035238
Maximum Volume	379.307	67.5176	0.258488	379.307	0.0171965	0.0117484	0.00699492
Average Volume	29.3099	11.9725	0.202941	33.7115	0.00901947	0.00437028	0.00497336
Invalid Volumes	0	0	0	0	0	0	0
0 ≤ V ≤ 0.001	906	0	0	296	0	610	0
0.001 < V ≤ 0.01	84554	12	0	46224	24	38262	32
0.01 < V ≤ 0.1	202053	14470	0	187571	8	4	0
0.1 < V ≤ 1	23851	5428	16	18407	0	0	0
1 < V ≤ 10	5035	0	0	5035	0	0	0
10 < V ≤ 1e+02	52894	7624	0	45270	0	0	0

- Here introduces the methods to check the mesh numbers

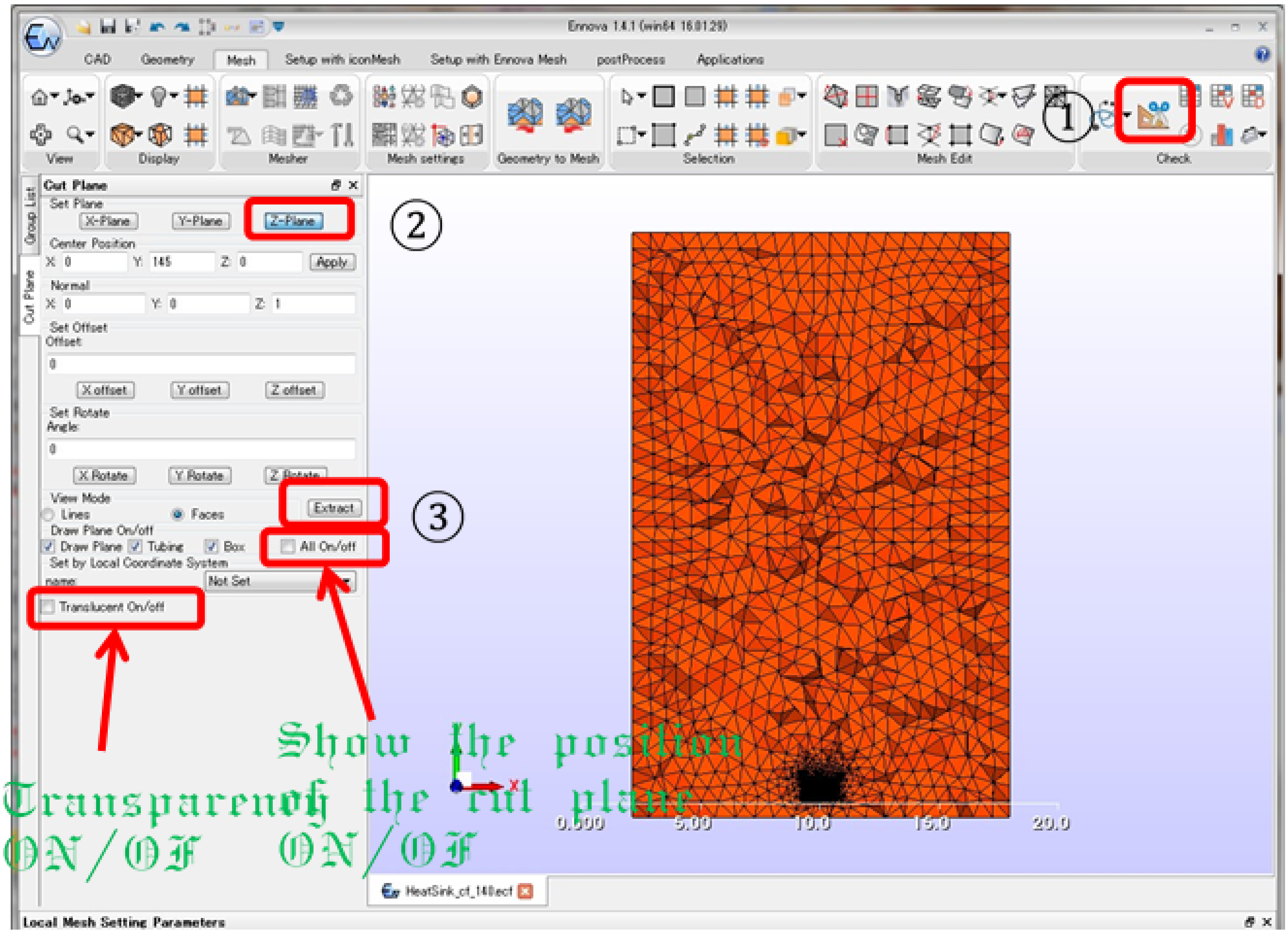


(1) Click ll icon

①: the total number of cells
②: triangular mesh (surface mesh)
③: square mesh (surface mesh)
④: tetrahedral mesh (volume mesh)
⑤: pyramid mesh (volume mesh)
⑥: pentahedron mesh (volume mesh)
⑦: hexahedral mesh (volume mesh)

Display a cut plane

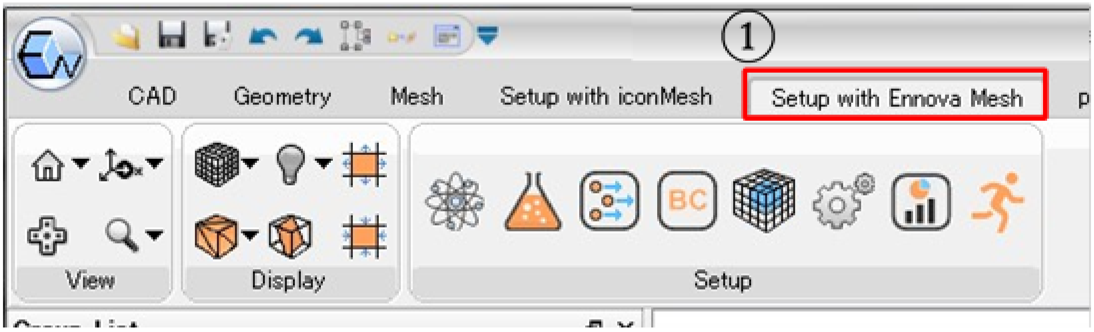
- Here we introduce how to display a cut plane



36

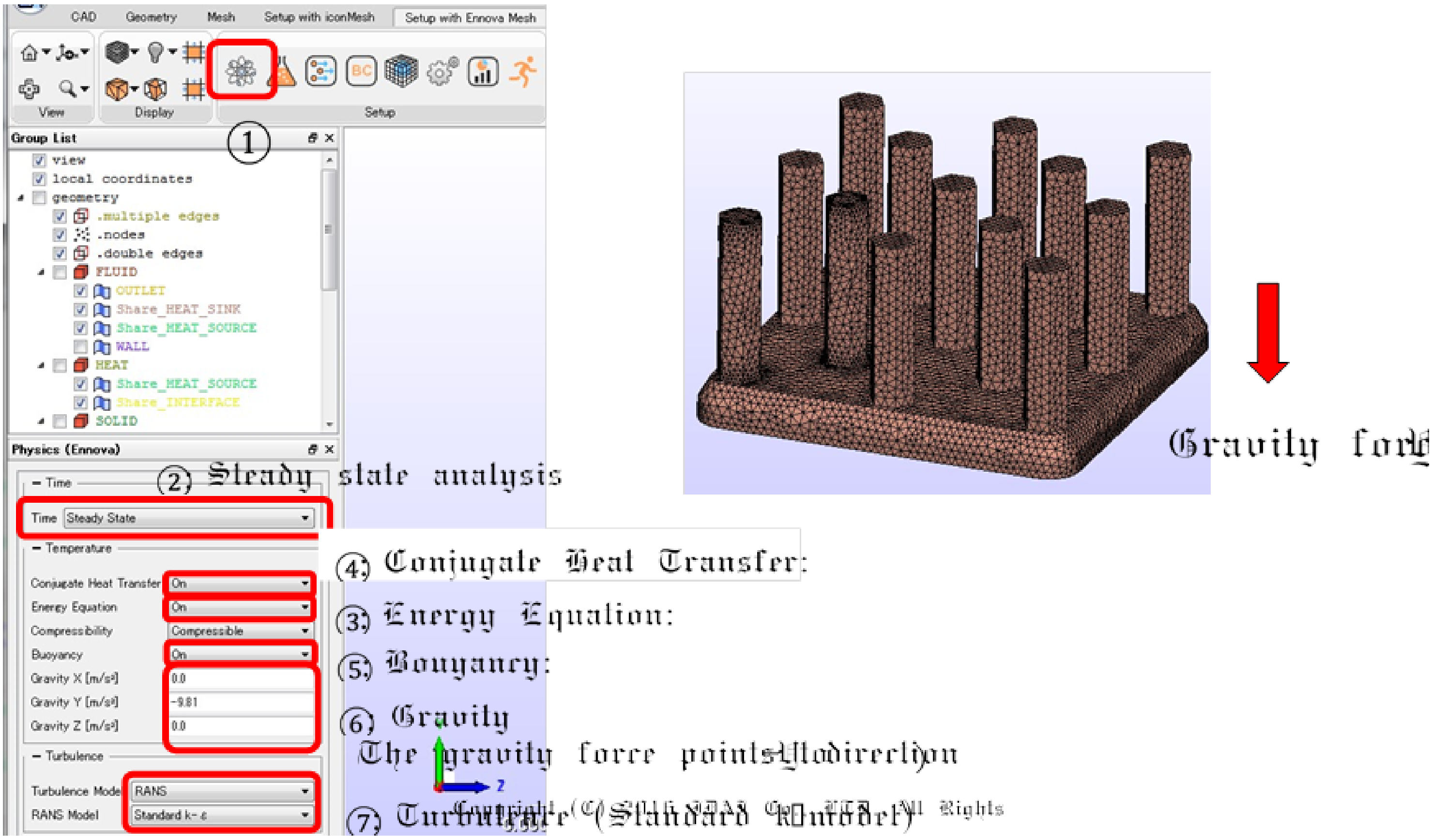
Settings for the solver

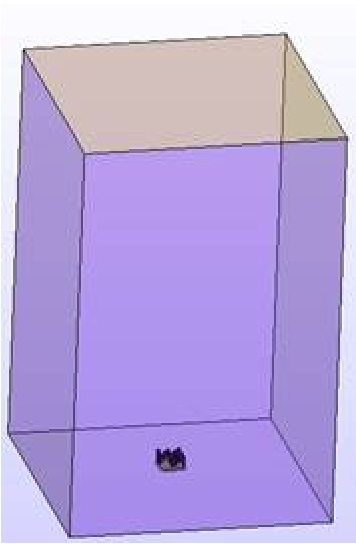
- Once the volume mesh has been generated with ennovaCFD, we can start the settings of the solver, which will be done by Setup with Ennova Mesh.



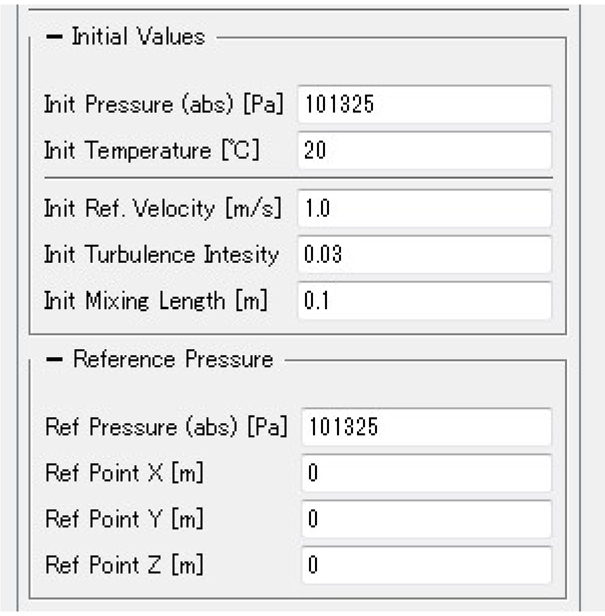
- Setting of the Physics

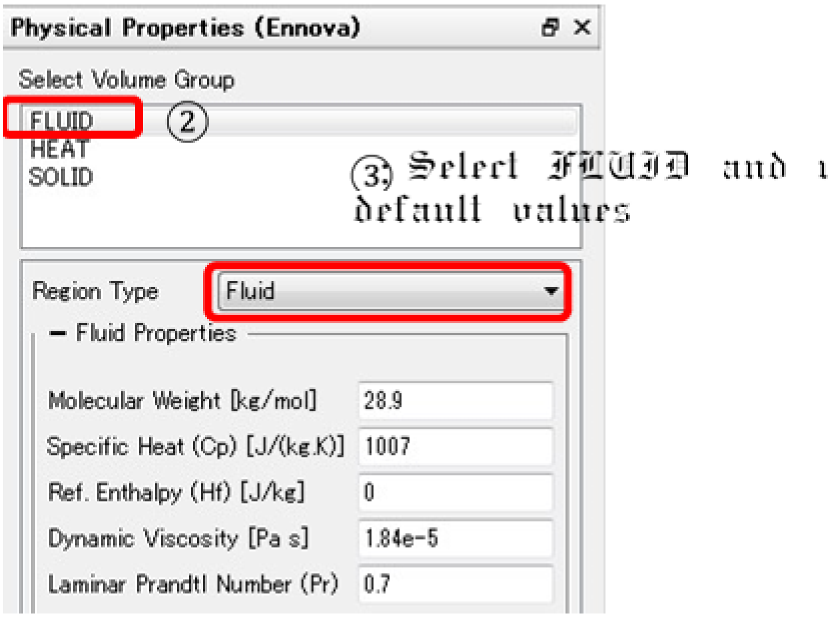
-





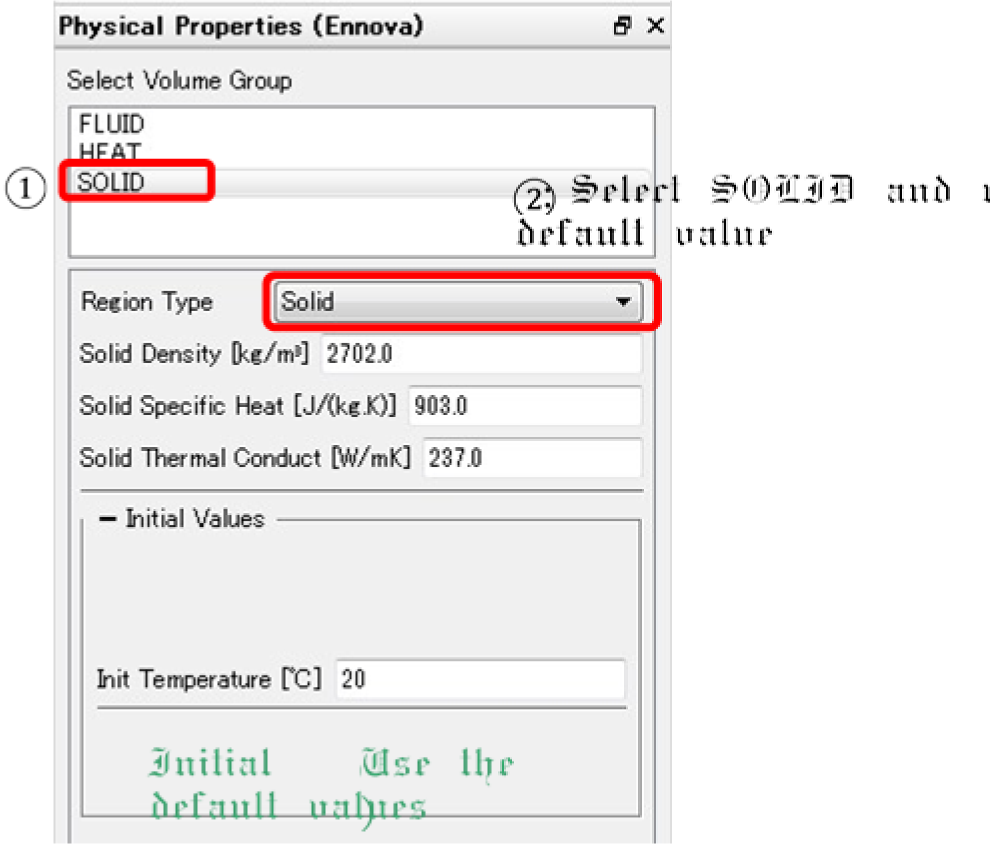
- Property settings (1/3)
- FLUID (outer domain) will be set to be air.

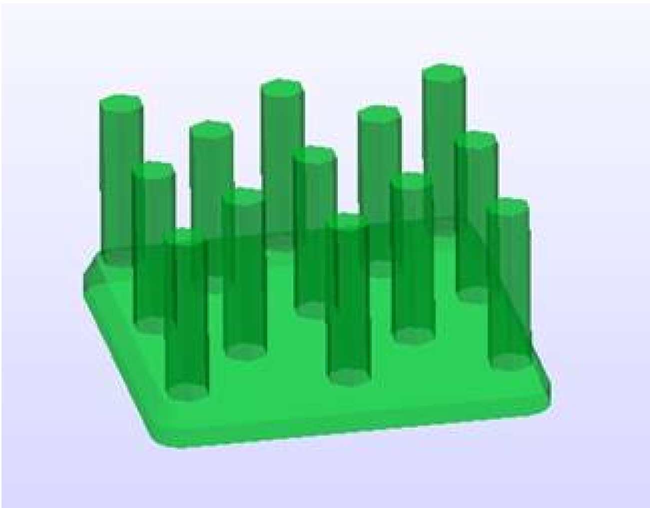




Initial values (Use the default values)

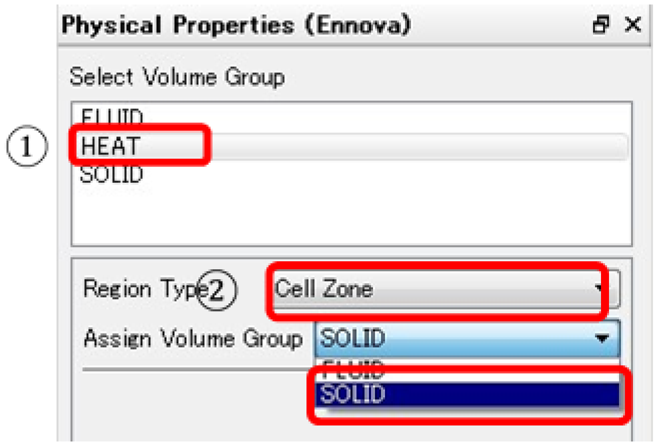
Property settings(2/3)

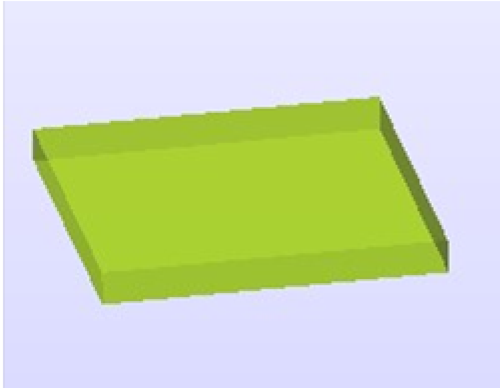


n 
SOLID (heat sink) will be set to be Aluminum.

Property settings (3/3)

n HEAT (heat source) will be set to be Cell Zone. It will have the same properties with the SOLID

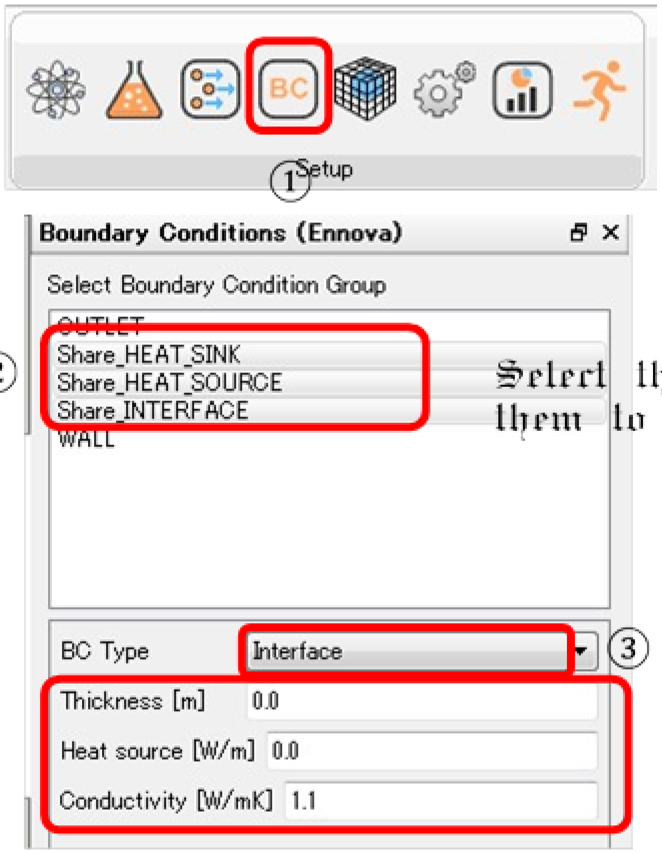




③: Select "SOLID"

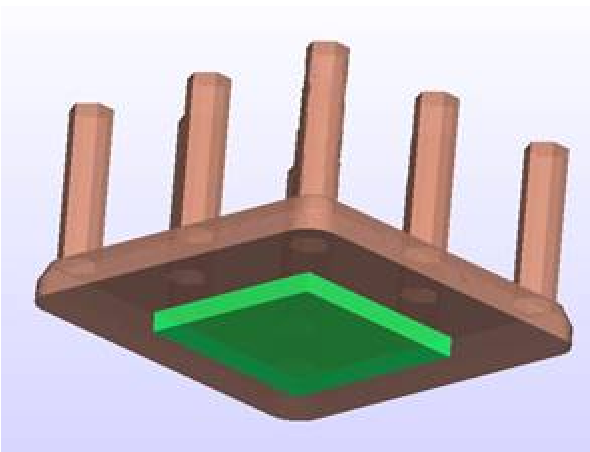
(HEAT belongs SOLID, i.e., they have the same physical properties.)

Setting the Boundary Conditions (1/3)



Select the Share~ bounda
them to be

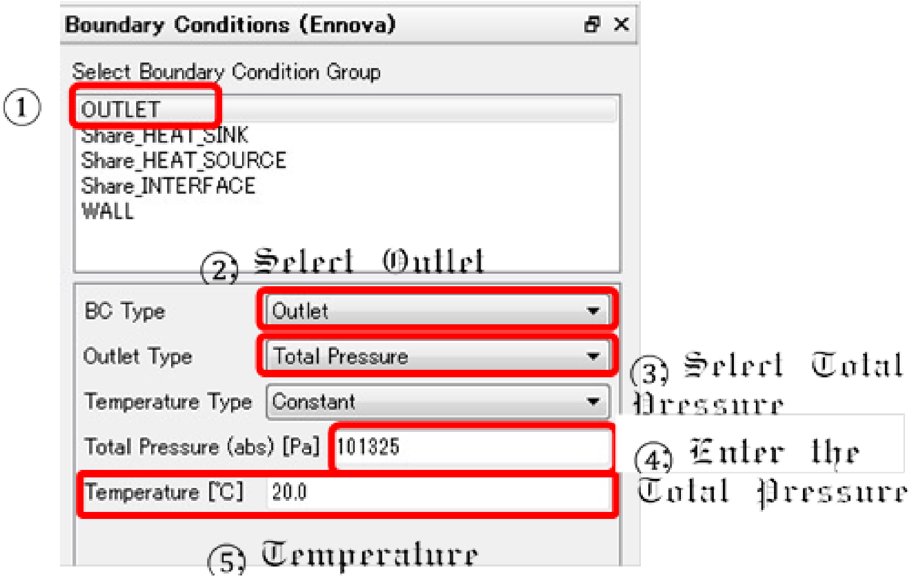
- ④ Use the default value

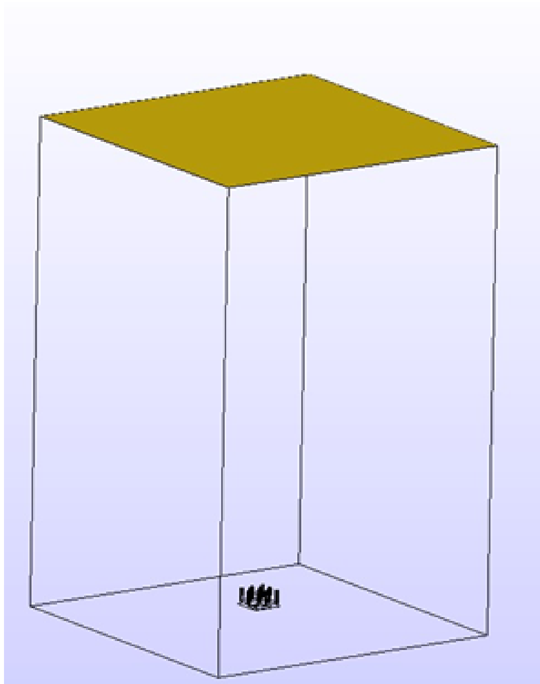


Set the boundaries starting with Share~ to be Interface.

Setting the Boundary Conditions (2/3)

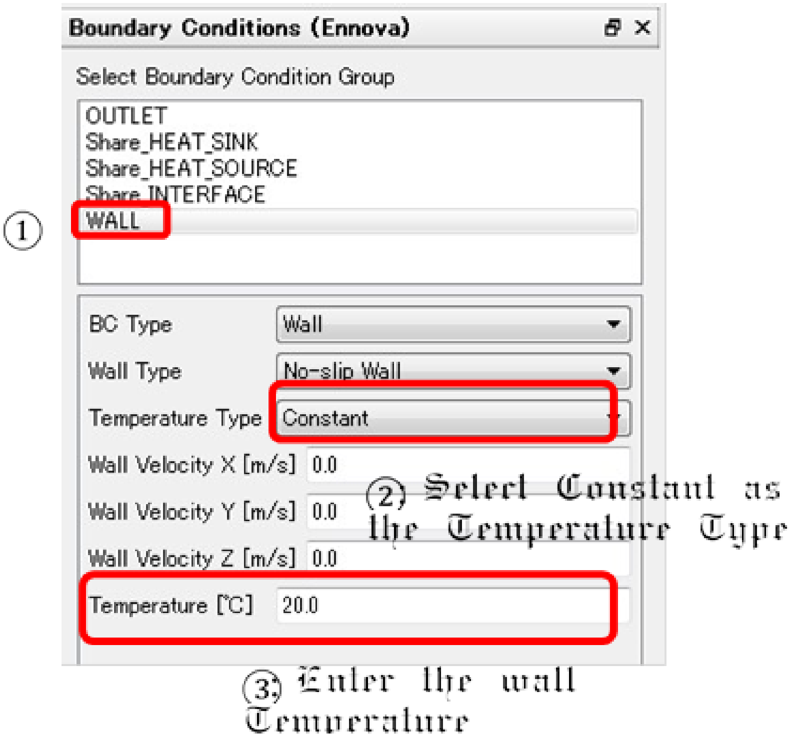
- OUTLET will be set to Total Pressure

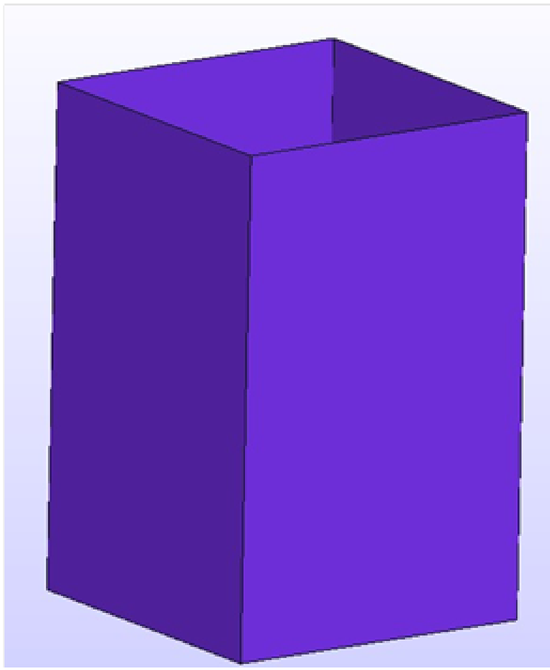




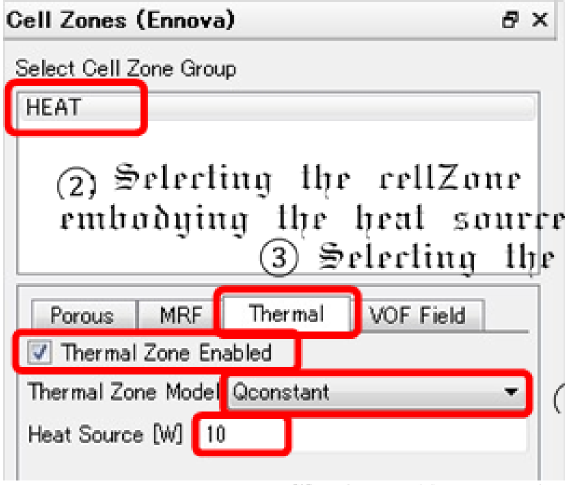
Setting the Boundary Conditions (3/3)

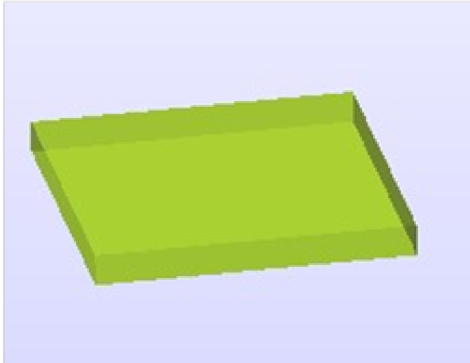
- WALL will be set to No-slip Wall with Constant Temperature.





Setting the Heat Source





(2) Selecting the cellZone embodying the heat source.

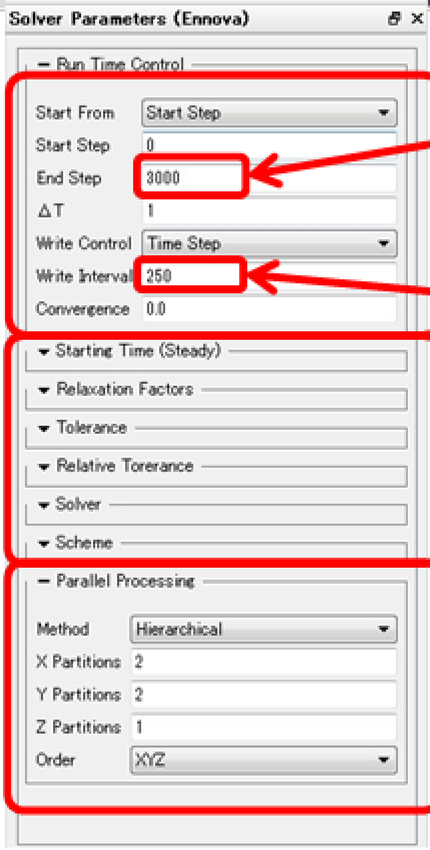
③ Selecting the Thermal tab

(4) ON

(5) Select Qconstant

(6) Enter the value of the Heat

Setting the Solver parameter



(2) Change to

See the next p

(4) Setting the parallel process properties

③: Change to 250

A) Start From
   - Select the start method

B) Start Step

- Start iteration

C) End Step
- End iteration

D) ΔT
- Timestep (1 is used as default for steady state analysis)

E) Write Interval
- Interval of Writing the output to hard disk

F) Convergence
- The criteria for convergence (When set to 0, the simulation will end until the specified iterations are practiced).

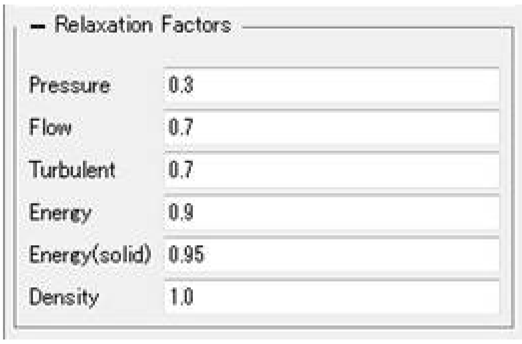
In this manual, except for ②~④

, default values will be used.

Setting the relaxation parameter

- In this manual, we use the default values.

A) Pressure
B) Flow
C) Turbulence
D) Energy (Fluid)
E) Energy (Solid)
F) Density

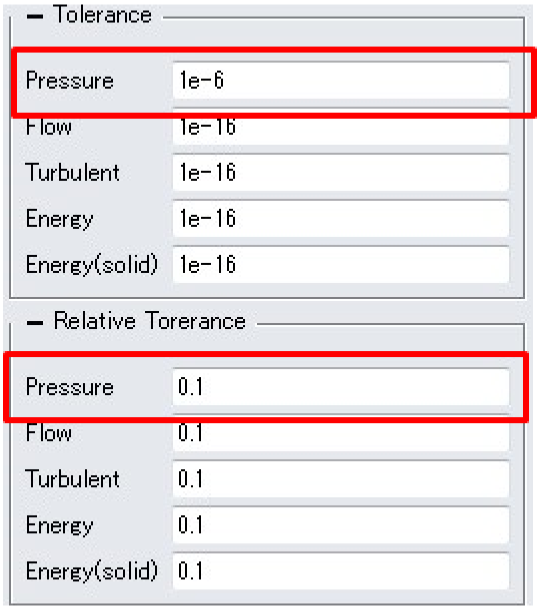


A B C D

E F

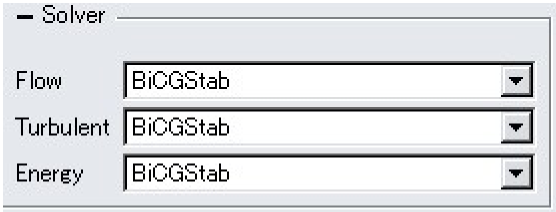
Setting the Tolerance/Relative Tolerance

Tolerance/Relative Tolerance of Pressure is set as follows.

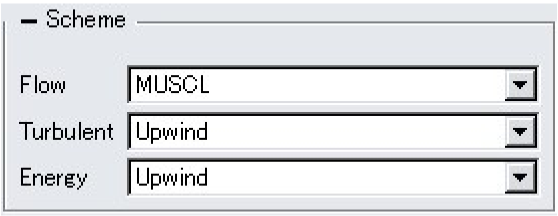


Setting the algebraic solvers

- Use the default values in this value .

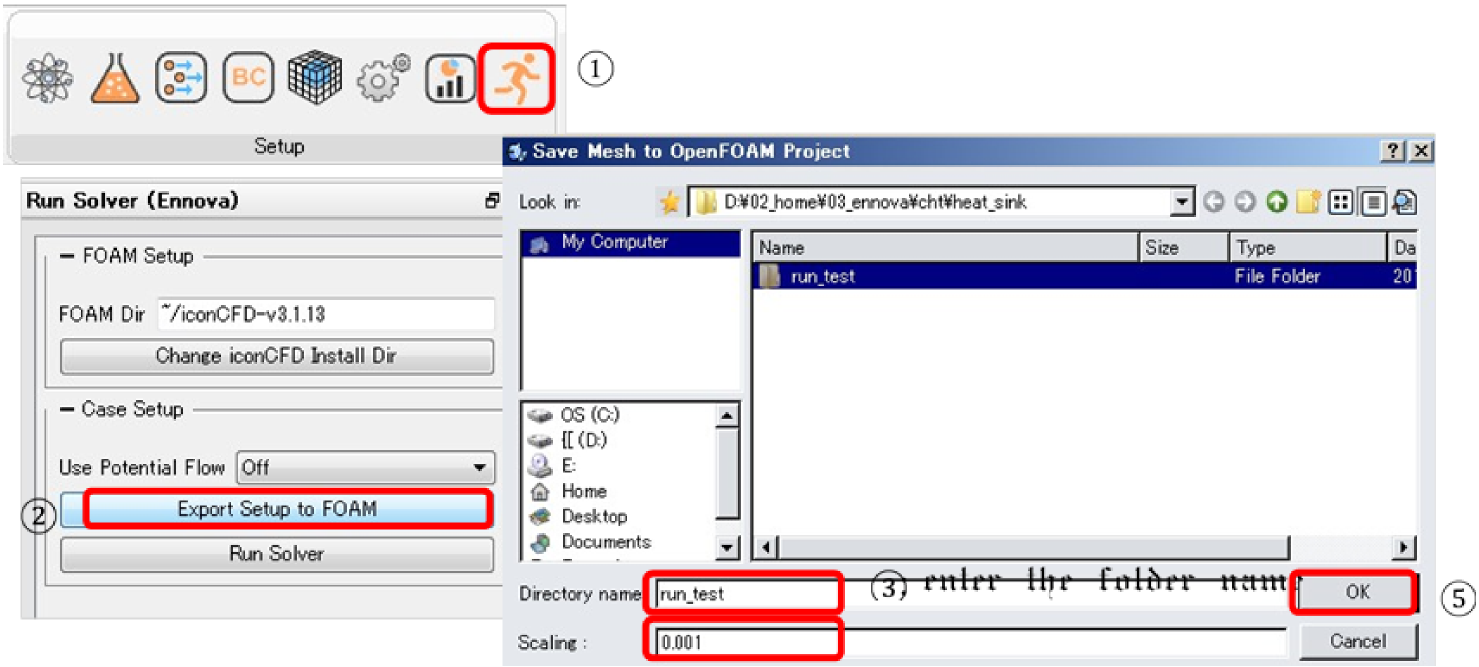


- Setting the differencing schemes
- Use the default values in this manual .



Exporting the iconCFD solver files

- The mesh data and the solver files will be written to the hard disk.



④ This model is created with unit of mm, it 1000 times to change the unit to be

Running the Simulation

After the mesh and solver settings in the Windows terminal, copy the files to

a system where iconCFD could run (such as simulation server).

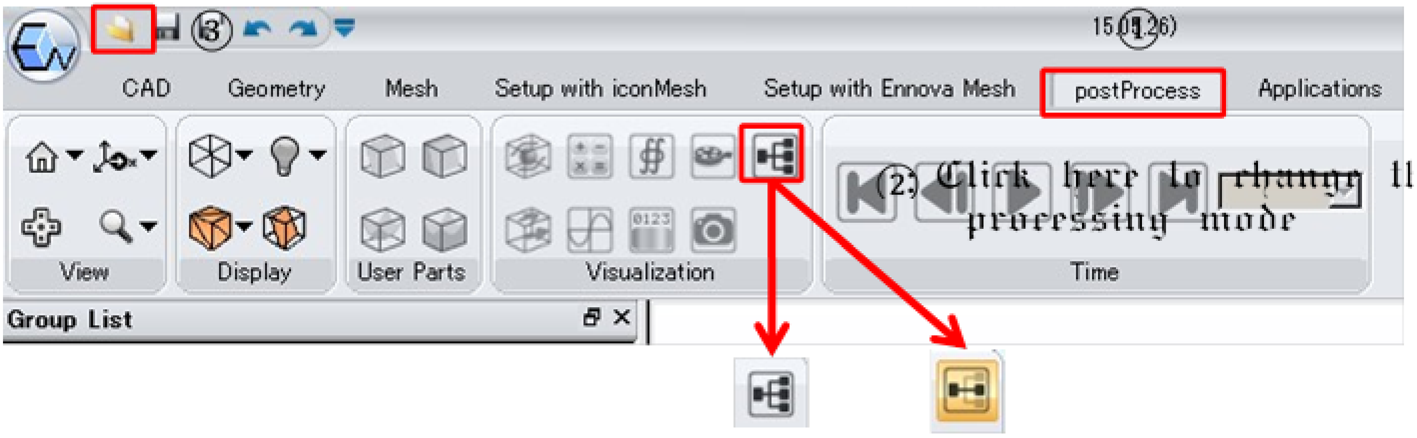
n Execute runHPC.sh to start simulations.

n Reading the simulation data from iconCFD（1/3）

n In the case of multiple processors

n Put ***system***， ***constant***， ***processor****， ****.foam*** files to a location accessible by Windows-PC.

n



Multiple processors　Single processor

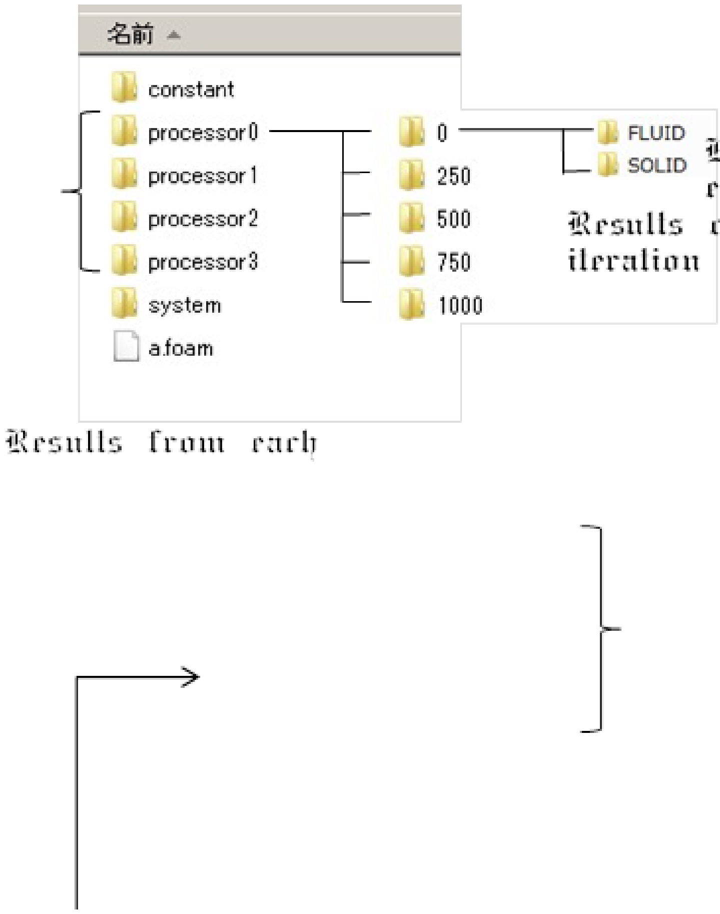
In the case of single processor

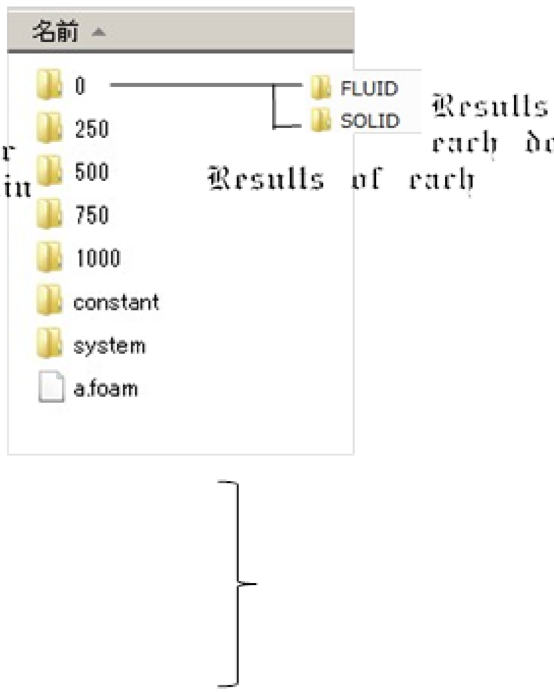
n Put ***system***， ***constant***， ***time directories***， ****.foam files*** to a location accessible by Windows-PC.

n Selecting the postProcessing method

n Before reading the files, choose to process the results using multiple processors or single processor.

n Reading the simulation data from iconCFD（2/3） n



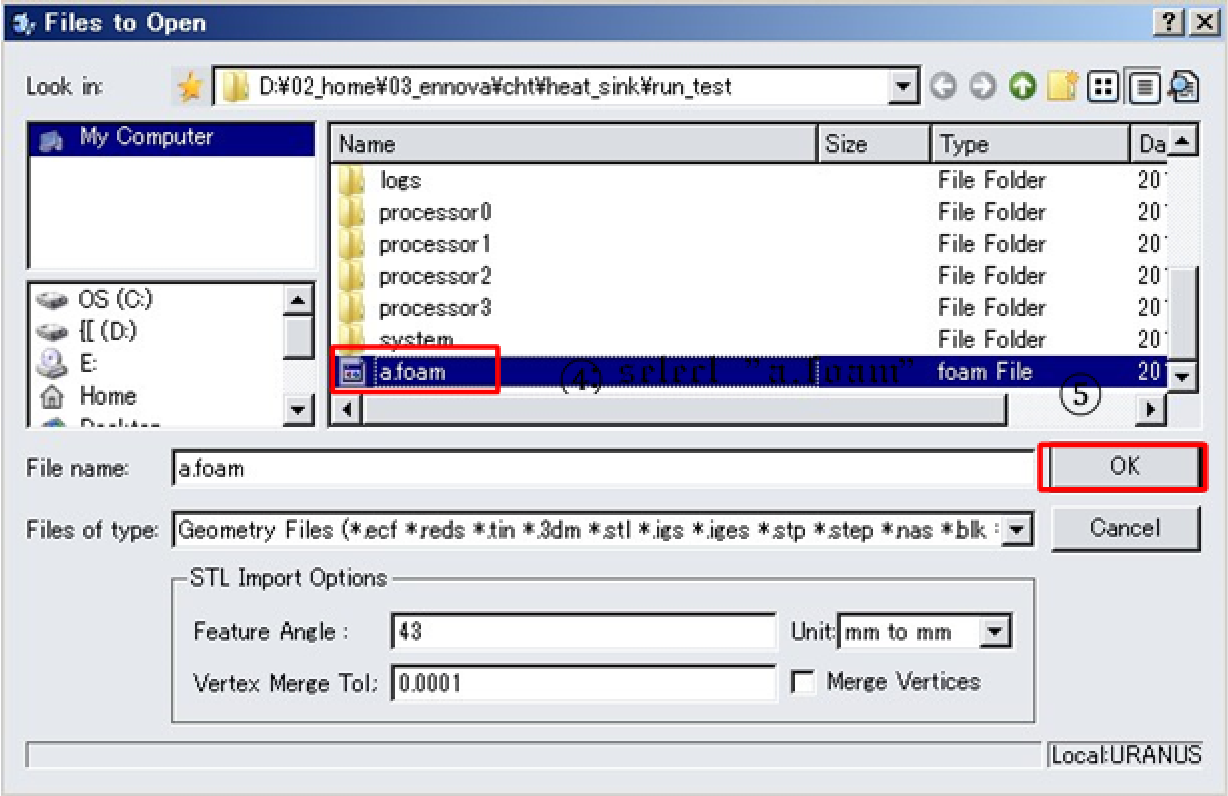


According to whether multiple or single processor(s) are(is)employed, these files are generated by the specified frequency (set in the Solver Parameter, p. 46) .

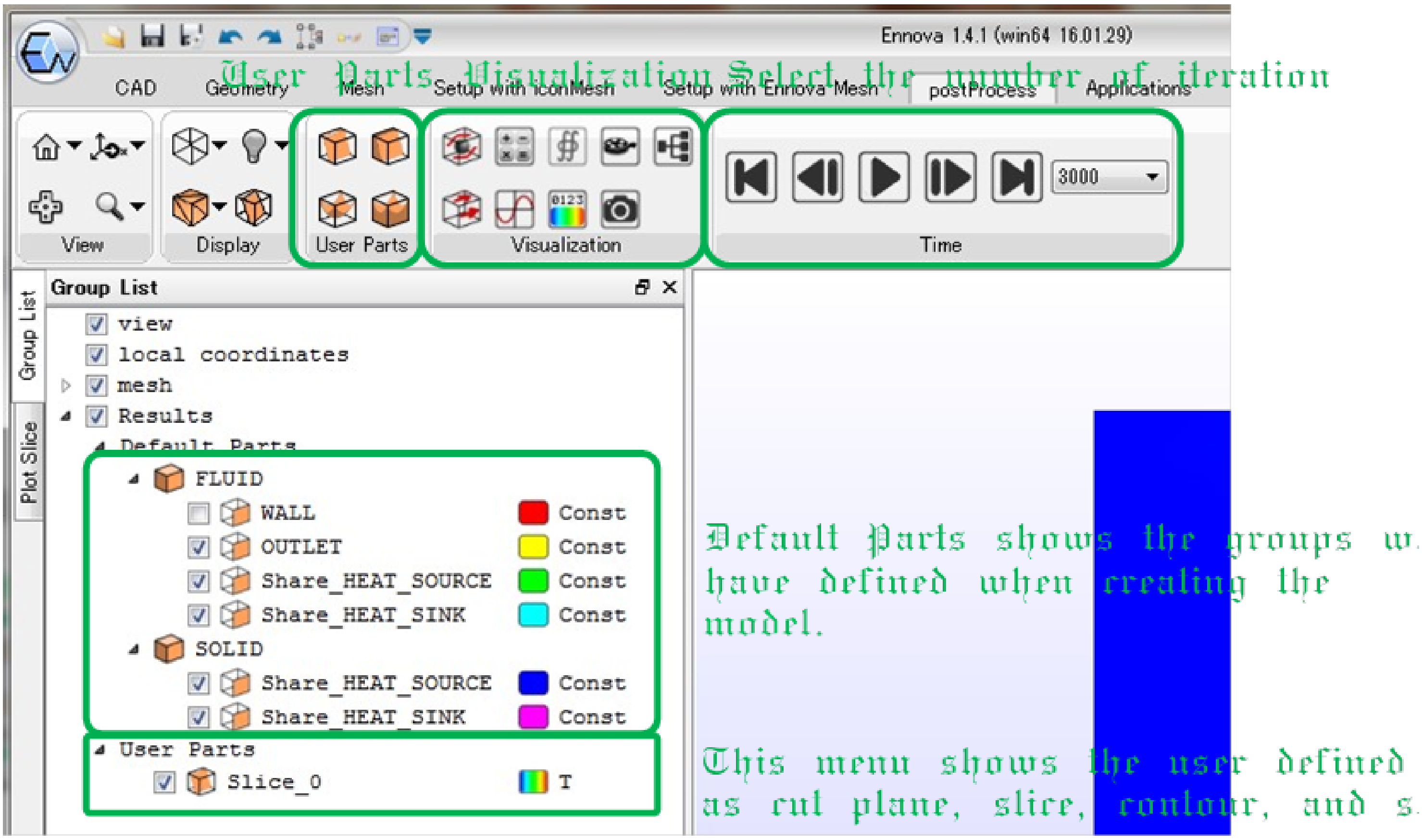
- n In the case of multiple processors n In the case of single processor

Reading the simulation data from iconCFD (3/3)

- n The simulation data could be read via the file “a.foam” in the folder. ”a.foam” is automatically generated using runHPC.sh.

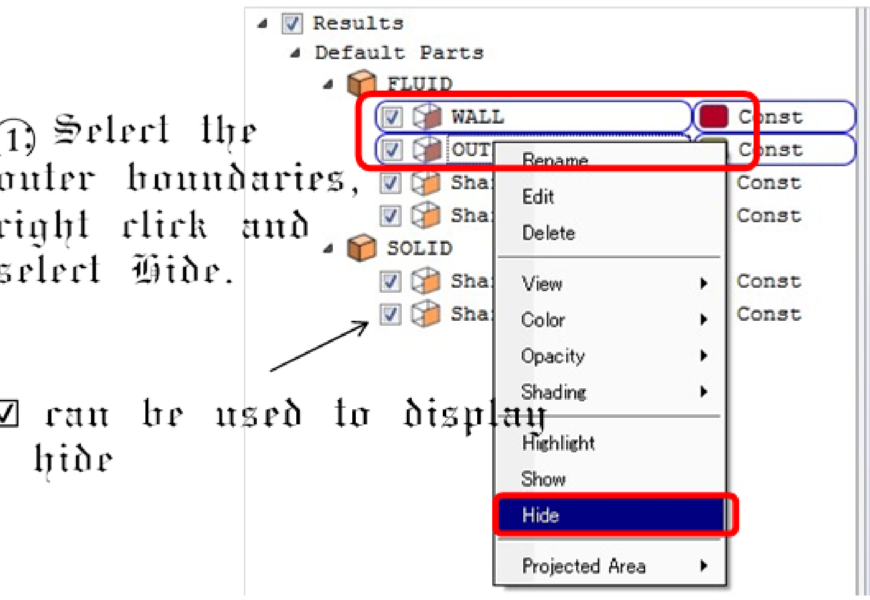


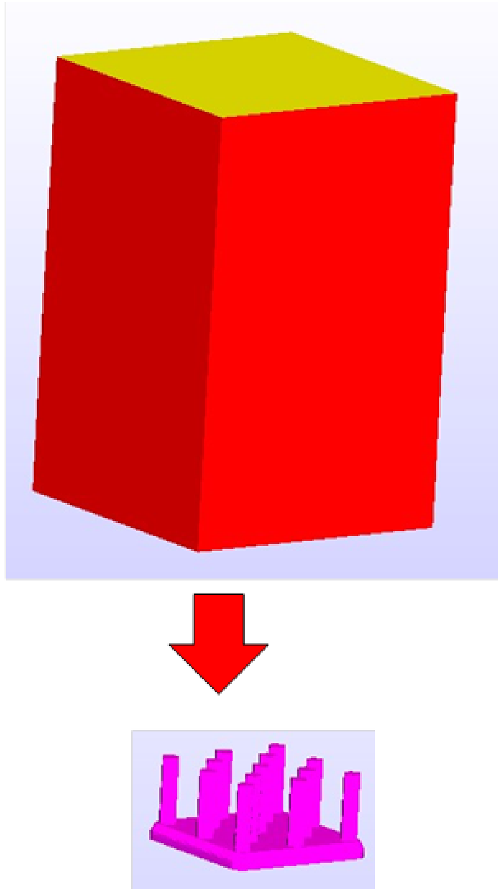
An introduction to the GUI of postProcessing



Display and hide of the group

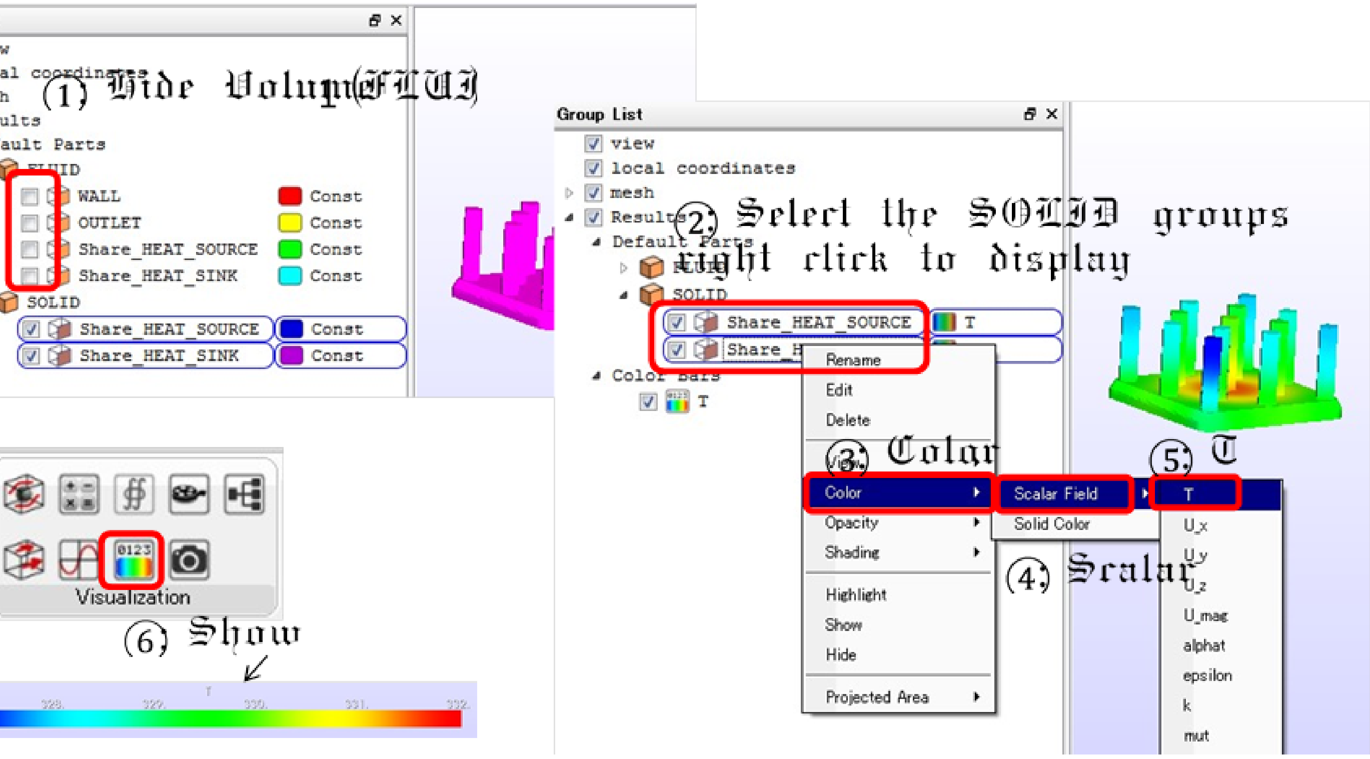
- Hide the outer boundary walls





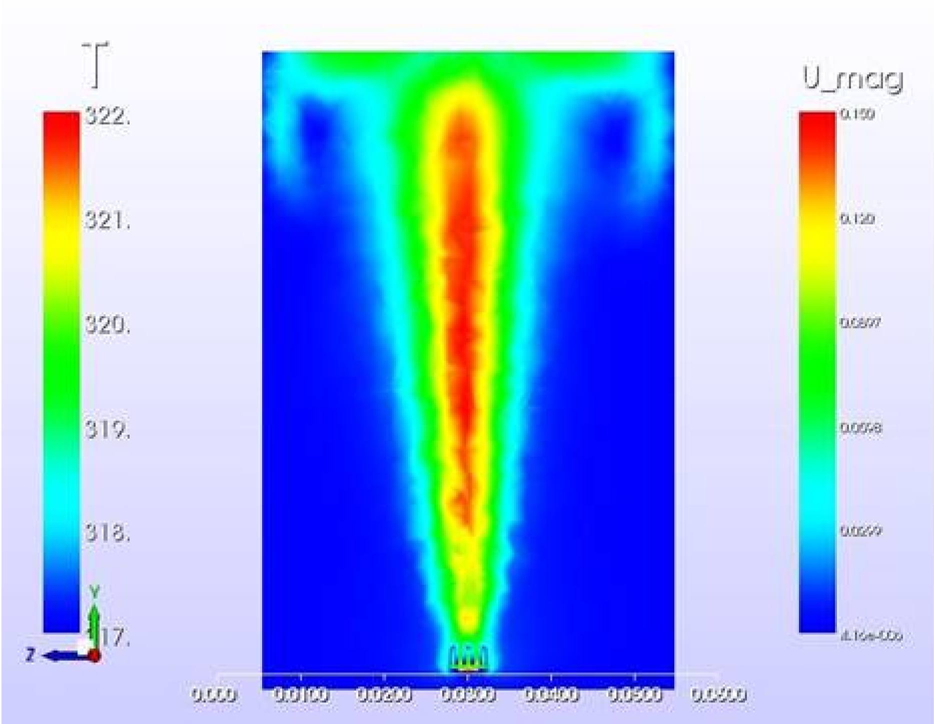
② Display the temperature

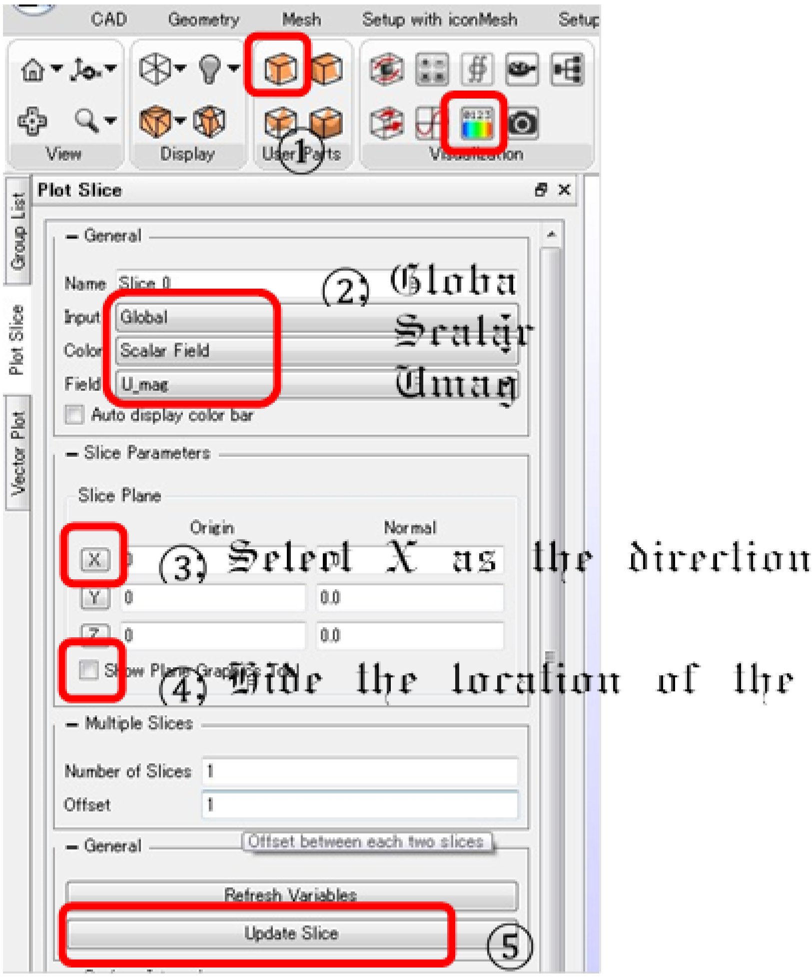
- The physical properties of the wall could be displayed on the group surfaces.



Display the velocity distribution on a slice

- Create a slice to show the velocity distribution

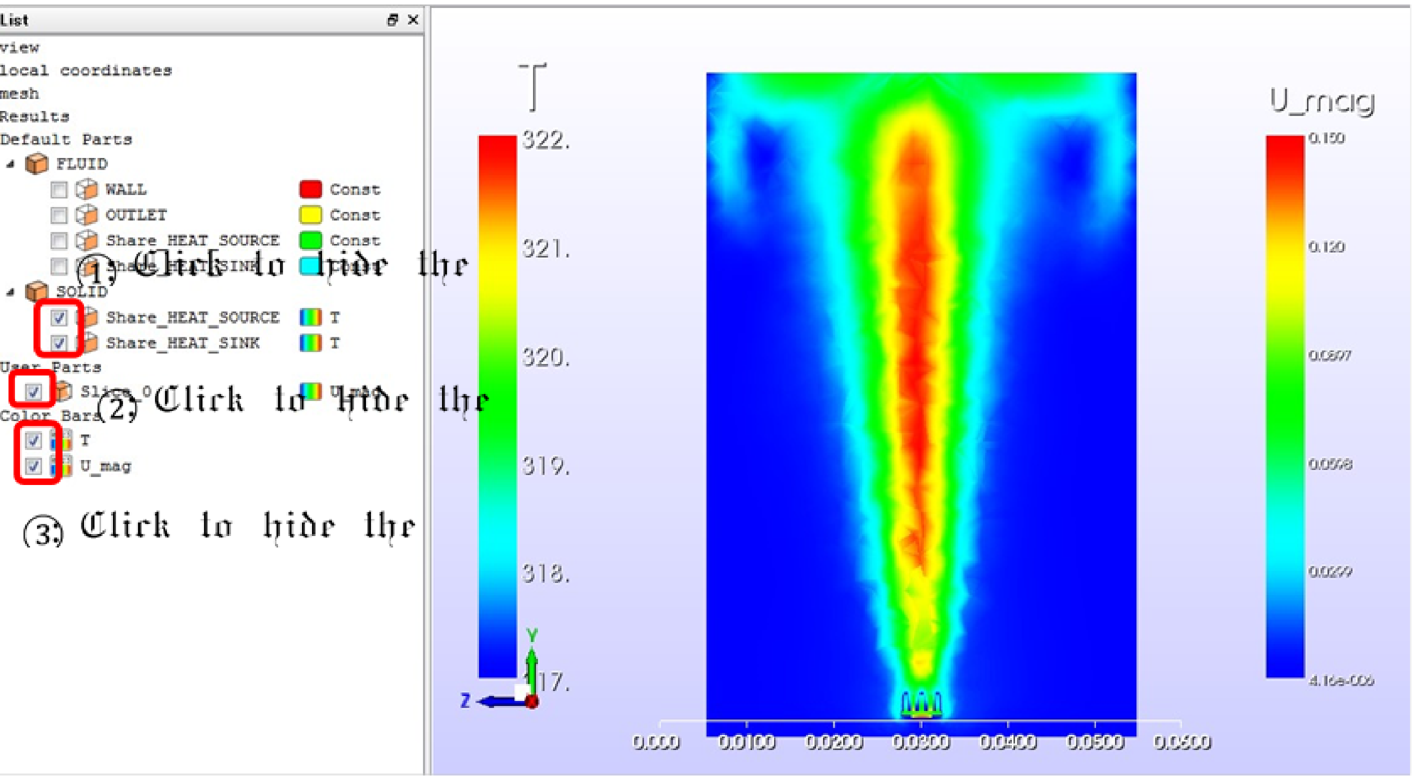




⑥:Display the colorbar

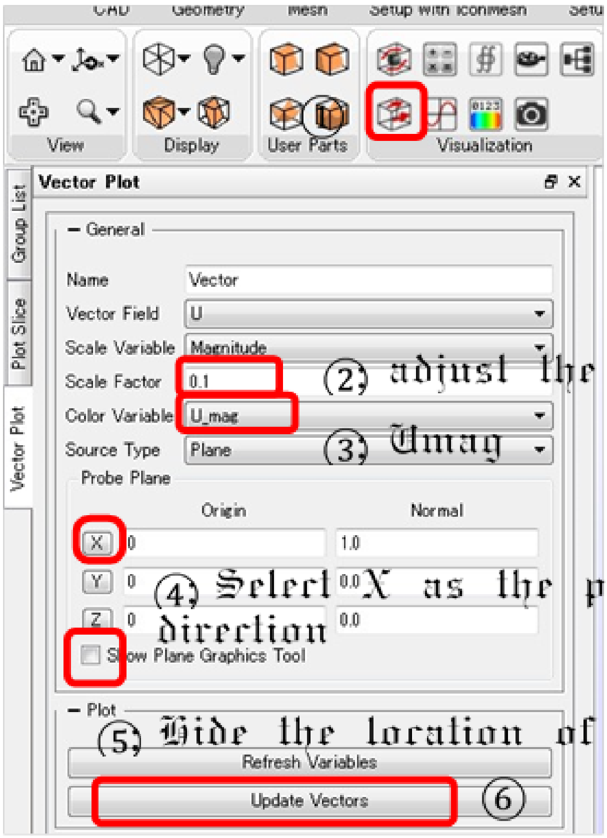
Contour map of temperature on solid+velocity distribution on the slice

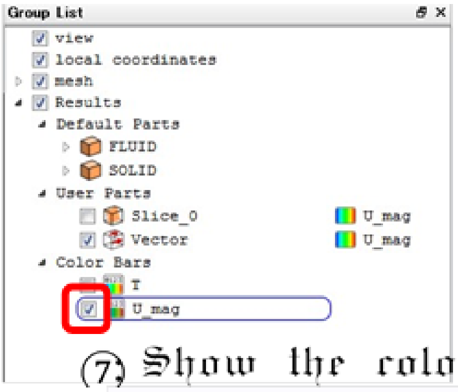
Hide the group, part and colorbar

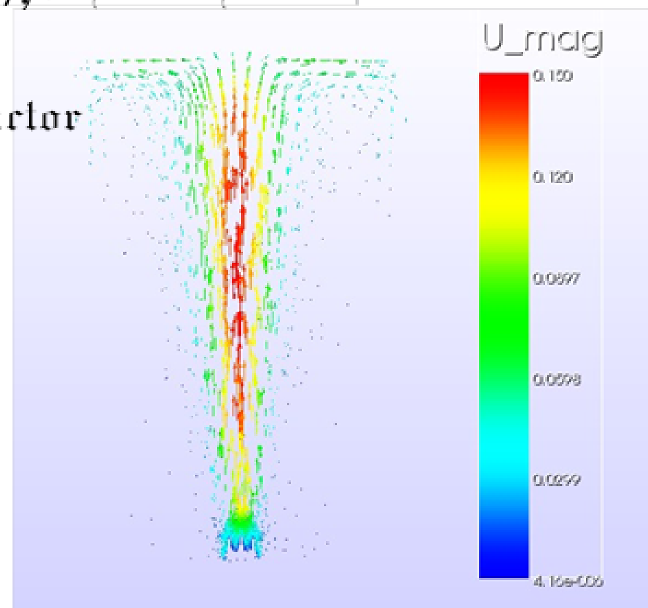


Display the velocity vectors

- Create vector plots







A manual for Wrapping In ennovaCFD v1.5

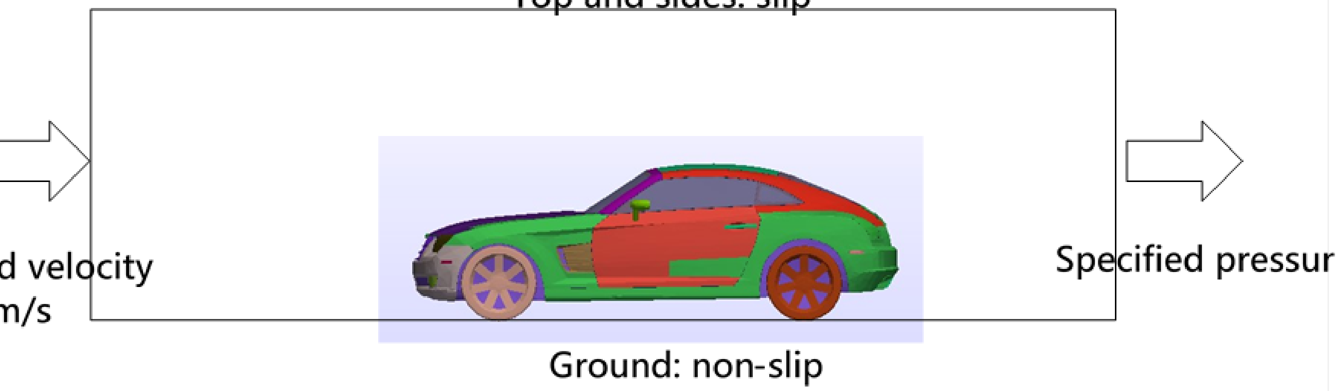
Contents

Introduction
Importing the STL data
Creating the computational domain
Changing the group names
Setting the wrapping mesh size
Setting the Thin cut
Setting the leak check
Saving the case data
Executing the surface wrapping and the results
0. Setting the iconCFD mesher
. Setting the iconCFD solver
2. Executing the simulation
3. Postprocessing

1． Introduction

This manual is for the incompressible steady state analysis for the flow around automobiles using ennovaCFD v1.5 and iconCFD 3.2.11.

The surface wrapping (shrinkWrap) of ennovaCFD and polyhedral mesher (iconHexMesh) of iconCFD is used to generate the mesh.



1． Introduction (contd)

The procedure of the manual is as follows.

Import STL data

↓

Surface wrapping settings

↓

Surface wrapping execution

↓

iconCFD mesher setting

↓

iconCFD solver setting

↓

iconCFD mesher・ solver execution

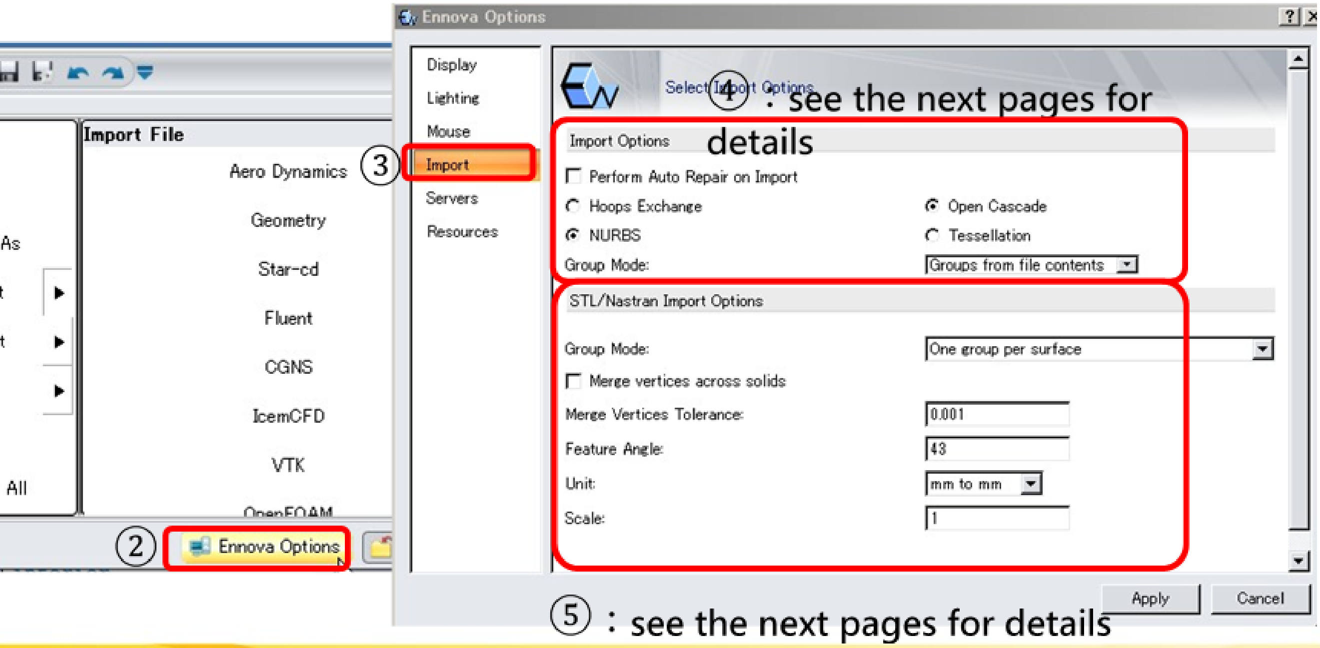
↓

postProcessing

2． Importing the STL data

Settings of import options (1/5)

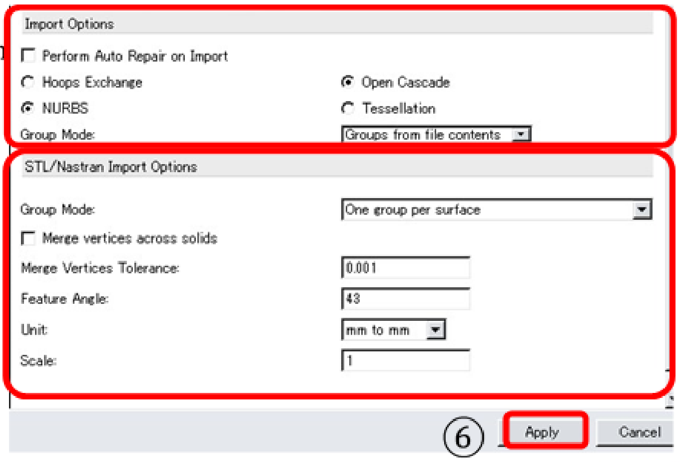
1. Click the Ennova icon to open the setting panel.
2. Open the Import tab and set the import options.



2． Importing the STL data (Contd)

Settings of the import options(2/5)

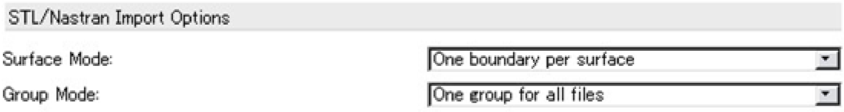
- Import Options
  - Perform Auto Repair on Import: Off
  - Open Cascade: On
  - Leave all the other setting as default.
- STL/Nastran Import Option (see the next pages for details.)
  - Group Mode One Group per surface
  - Merge vertices across solids off
  - Merge Vertices Tolerance 0.001
  - Feature Angle 43
  - Unit m to m
  - Scale 1



2． Importing the STL data (Contd)

Setting the import options (3/5)

- Explaination of ***STL/Nastran Import Option***



- Group Mode:
  - One group per file
    - When importing multiple STL files, create the geometry groups for every STL files.
  - One group for all files
    - When importing multiple STL files, create one geometry group for all the files.
  - One group per surface
    - Create geometry groups for every sub-part of the STL file.

2． Importing the STL data (Contd)

Setting the import options (4/5)

- Explanation of STL/Nastran Import Option

- Merge vertices across solids
  - When importing multiple files, merge the vertices within the tolerance.
  - This options is used when One group per file/One group for all files is selected.

- Merge vertices Tolerance.
  - Merges the gaps below this value.

2． Importing the STL data (Contd)

Setting the import options (5/5)

- Explaination of ***STL/Nastran Import Option***

Feature Angle:	43
Unit:	m to m
Scale:	1

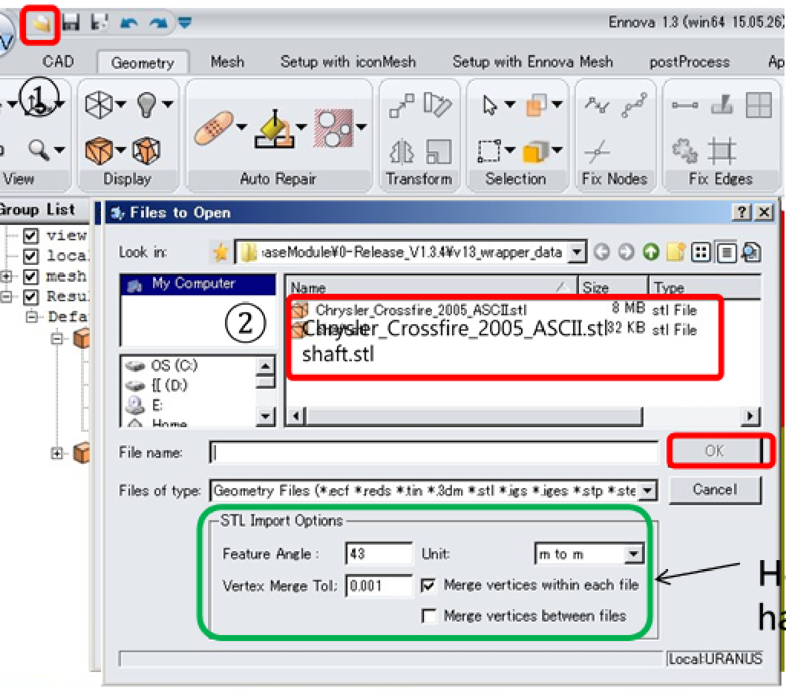
- Feature Angle
  - Set the minimum angle between polygonal facets when displaying feature lines.

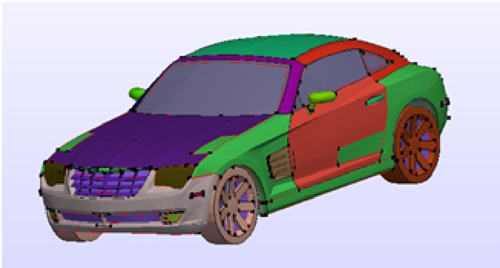
- Unit
  - Scale the model's size by unit.
  - Scaling with the same unit (e.g., m to m) does not change the size of the model.

- Scale
  - Scale the model's size with specified ratio.
  - No scale if specified to 1.

2． Importing the STL data (Contd)

Import the Chrysler_Crossfire_2005_ASCII.stl and shaft.stl files.



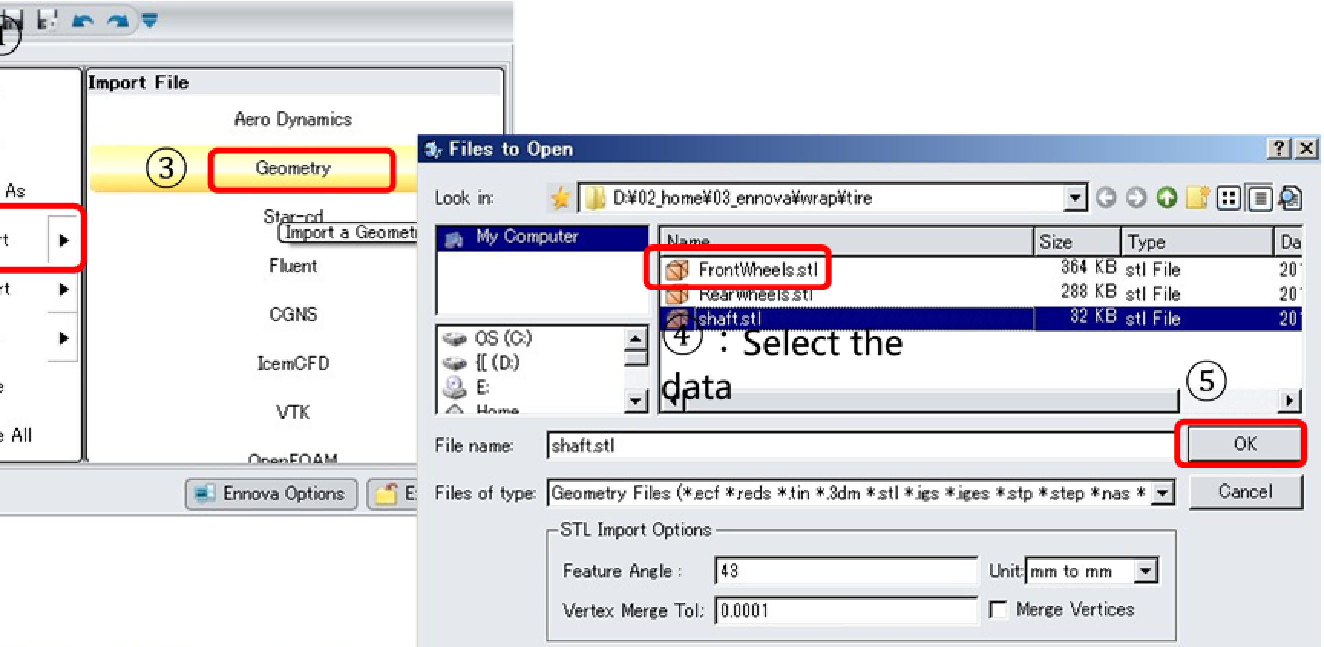


Here shows the options we have set in previous slides.

2． Importing the STL data (Contd)

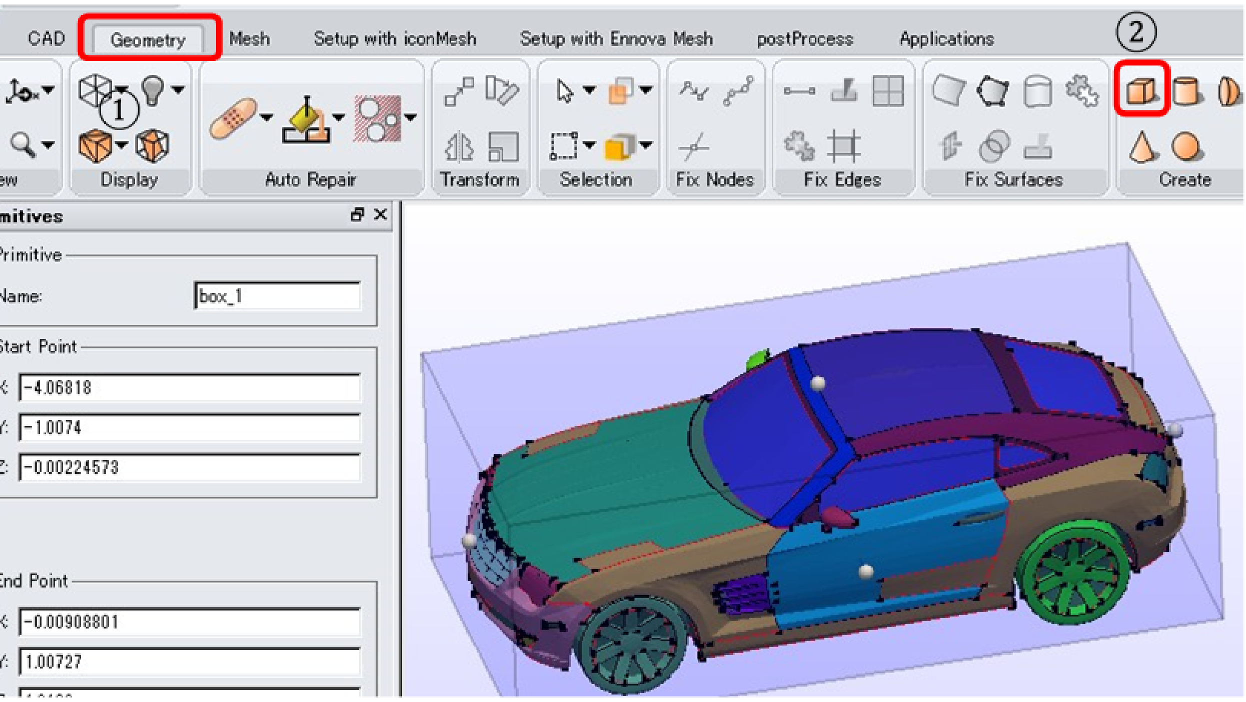
The method of add geometry data

- This slide shows how to add more data after the geometry has been imported for the first time. (not required for the current case)



3. Creating the computational domain

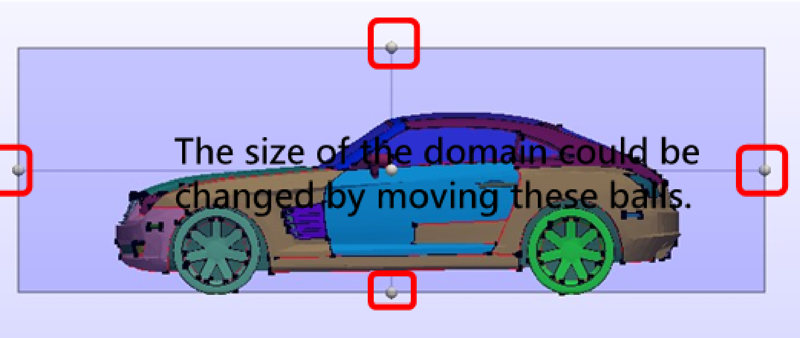
Using the primitive function to create the outer domain.

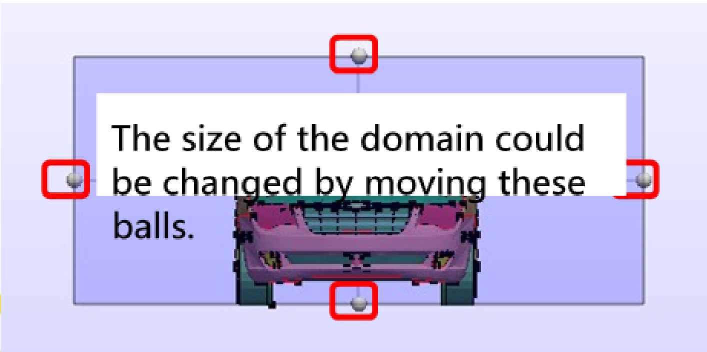


3． Creating the computational domain (contd)

Enter sizes, or move the balls on the GUI.

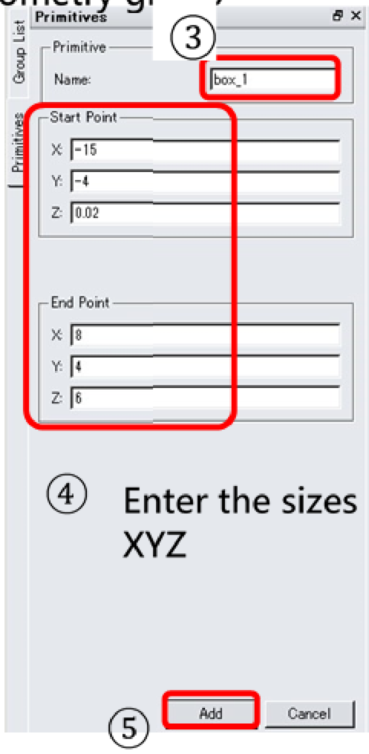
- In the current case, we enter the sizes to create the outer domain.





Enter the name of the geometry group

③



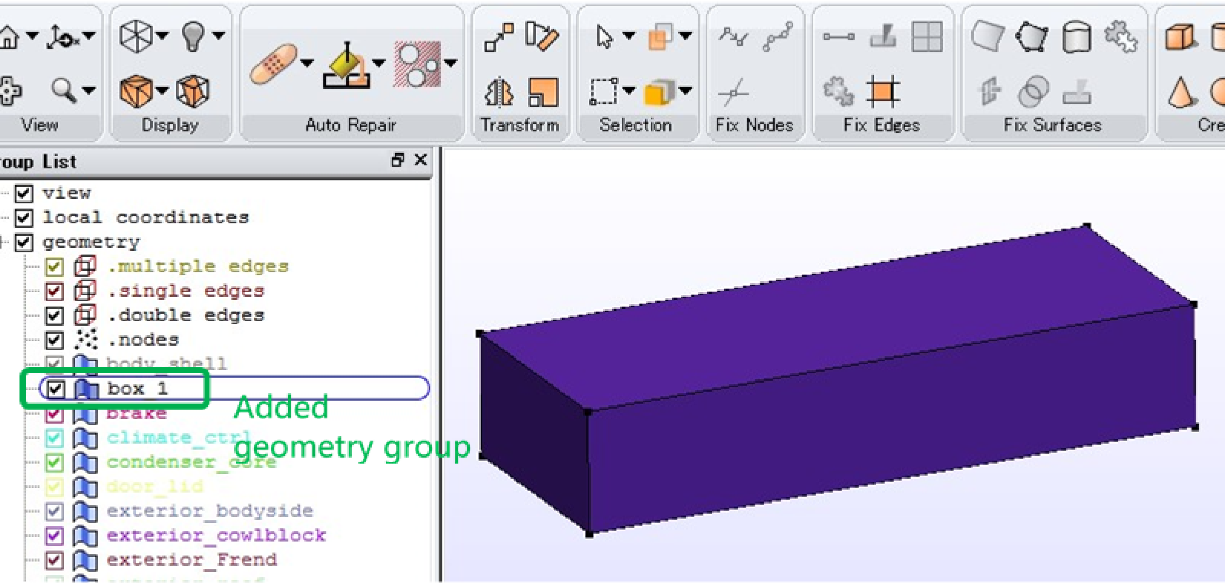
④ Enter the sizes at XYZ

⑤

3． Creating the computational domain (contd)

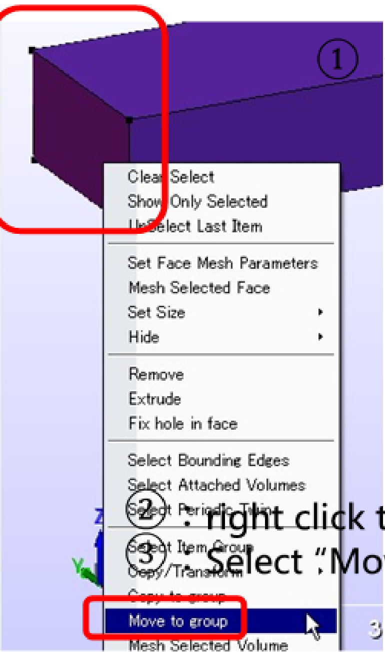
Confirming the outer domain.

- Upon completion, the new geometry group is added to the geometry tree.



4． Changing the group names

Changing the name of the inlet.



① : Click this surface

② : right click to show the menu

③ : Select "Move to group"

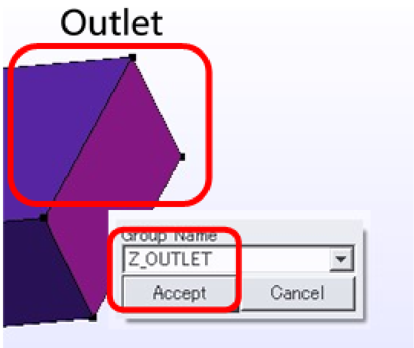
④ : Enter the new name

⑤ : Click "Accept"

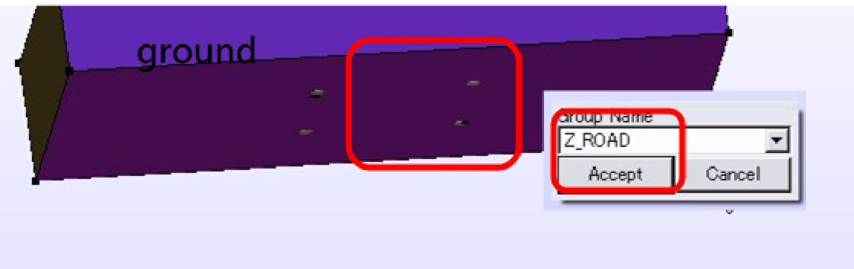
NOTICE: In the case of iconCFD mesh, do not use numbers as the first letter of the group name since it could cause error in the iconHexMesh.

4． Changing the group names (Contd)

The same as the inlet, change the group names for the outlet and the ground.

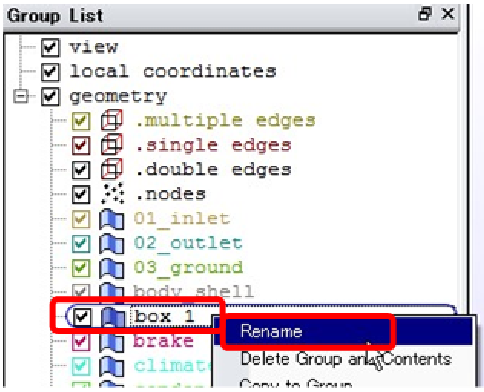


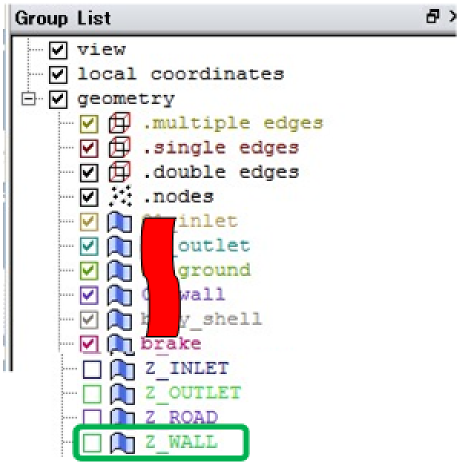
Geometry tree after name changing.



4． Changing the group names (Contd)

Change name of the remaining of box_1 to Z_WALL.



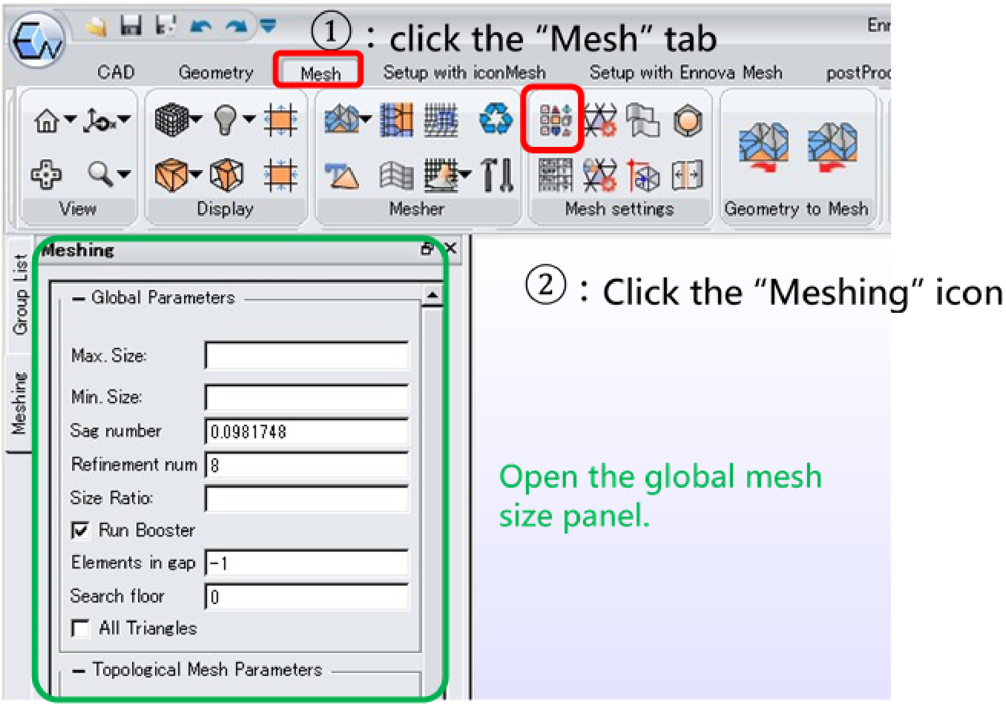


①：Click on the item whose name needs changing
②：Right click to show the menu
③：Select "Rename"
④：Enter the new name

5． Settings of the wrapping mesh size

Setting the mesh size for the surface wrapping.

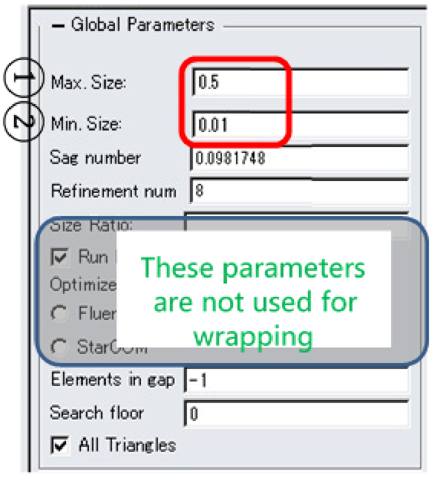
- Setting the global mesh size



5． Settings of the wrapping mesh size (Contd)

Details of the global mesh size panel (1/3)

- In the current case we use the following values.



①：Max　Size: 0.5

Maximum mesh size for the model

②：Min Size: 0.01

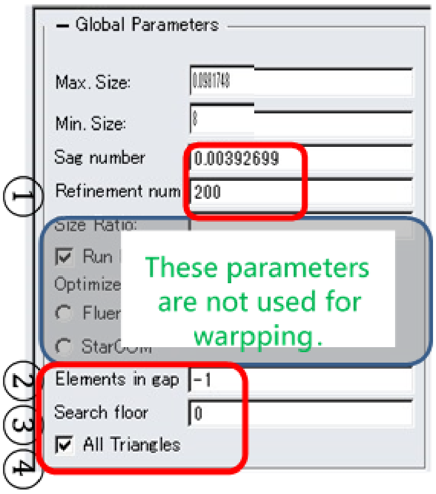
Mimimum mesh size for the model

Note： In ennovaCFD 1.5, the local mesh size can not exceed the maximum mesh size specified above. So, the maximum mesh size should be set based on the scenario that generate the largest meshes. Later, the mesh size could be refined using the local mesh setting.

5．Settings of the wrapping mesh size (Contd)

Details of the global mesh size panel (2/3)

- In the current case we use the following values.



①：Refinement num: 200

fit ratio for the mesh on the curve surfaces

See P22 for more details

②：Elements in gap: -1 (default)

See P23 for more details

③： Search floor: 0 (default)

See P23 for more details

④：All Triangles: On

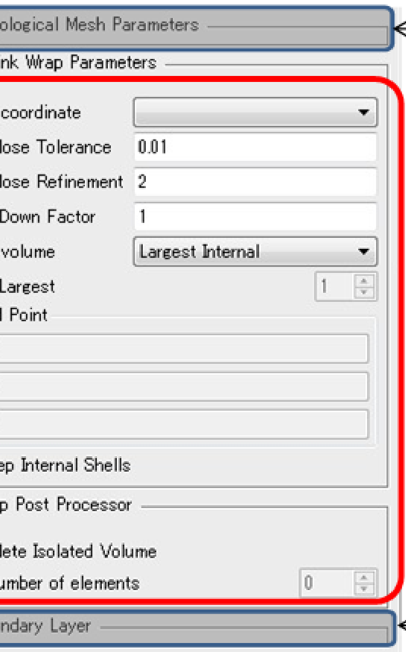
If this option is turned on, the surface mesh will all be triangles.

If this option is turned off, rectangular meshes will be created where possible, the other places will be meshed with triangles.

5． Settings of the wrapping mesh size (Contd)

Details of the global mesh size panel (3/3)

- In the current case we use the following values.



Not used in wrapping

①：specify the wrapping coordinate (unavailable in v1.5)
②：gap close tolerance (gaps smaller than the value will be filled in)
③：meshes in the gaps that are filled in
④：scale factor of the wrapped mesh size
⑤：setting of wrapping extraction location
⑦：maintenance of the baffle element
⑧：removal of the microelements generated during wrapping
See the next pages for detailed information

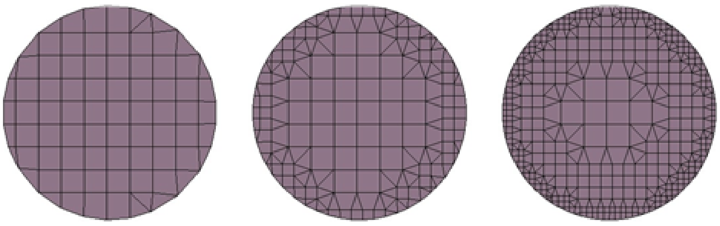
Not used in wrapping

5． Settings of the wrapping mesh size (Contd)

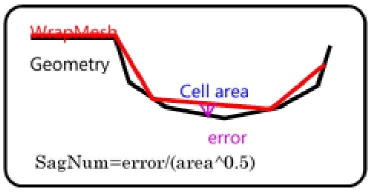
Explanation of Refinement Number and Sag Number

Refinement Number
The estimated value of nodes on a circle when splitting. The larger the value, the finer the mesh.

RefNum=16 RefNum=32 RefNum=64



Sag Number
Wrapping error divided by the squared root of the cell area. The lower the value, the finer the mesh



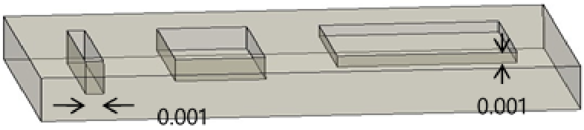
In addition, if one of the parameter is decided, the other one is also fixed.

Refinement Number＝π/Sag Number/4

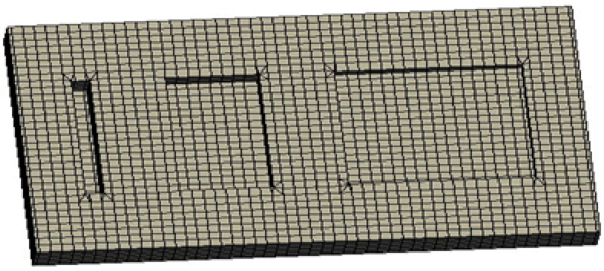
If the **Min. Size** is left blank, the minimum mesh size will be decided by the scale of the model.
Refinement is restricted by the specified minimum mesh size.

5． Settings of the wrapping mesh size (Contd)

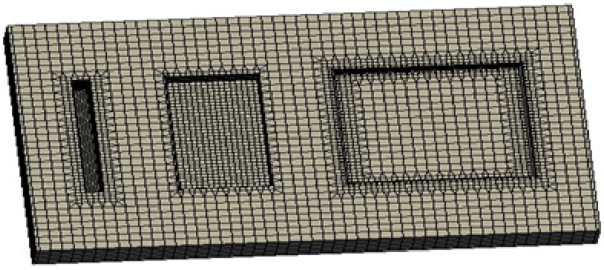
Explanation of Element in gap and Search floor



Example of settings
Max Size: 0.001
Min Size: blank



＜default value (no refinement)＞
Element in gap: -1
Search floor: 0



＜4 subdivisions within a length of 0.001＞
Element in gap: 4
Search floor: 0.001

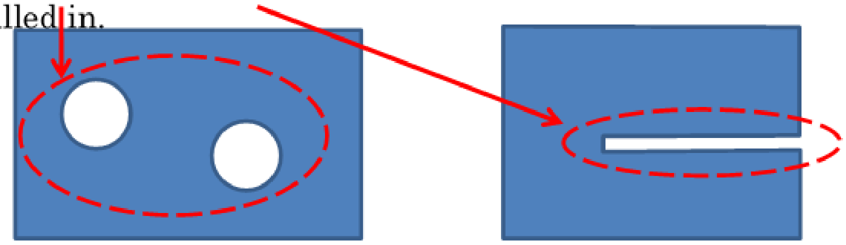
When Min. Size is left blank, the smallest mesh size is: 「Search floor ／Element in gap」

5． Settings of the wrapping mesh size (Contd)

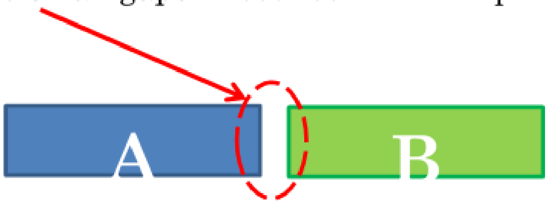
Explanation of Gap Close tolerance

- The gaps whose sizes are smaller than the gap close tolerance will be automatically filled in by the surface wrapping.

- The interior gaps whose size is smaller than the specified tolerance will be filled in.



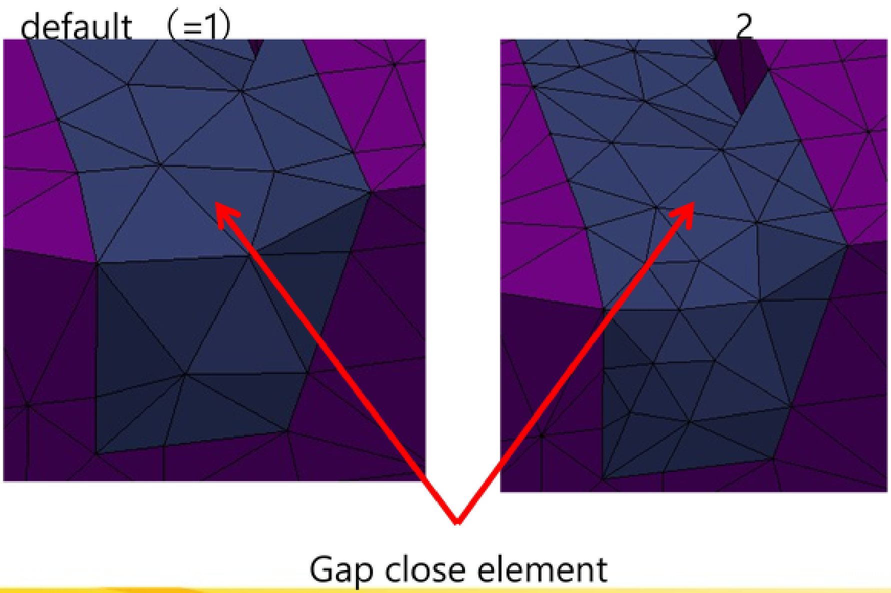
- The small gaps in between two components will be filled in.



5． Settings of the wrapping mesh size (Contd)

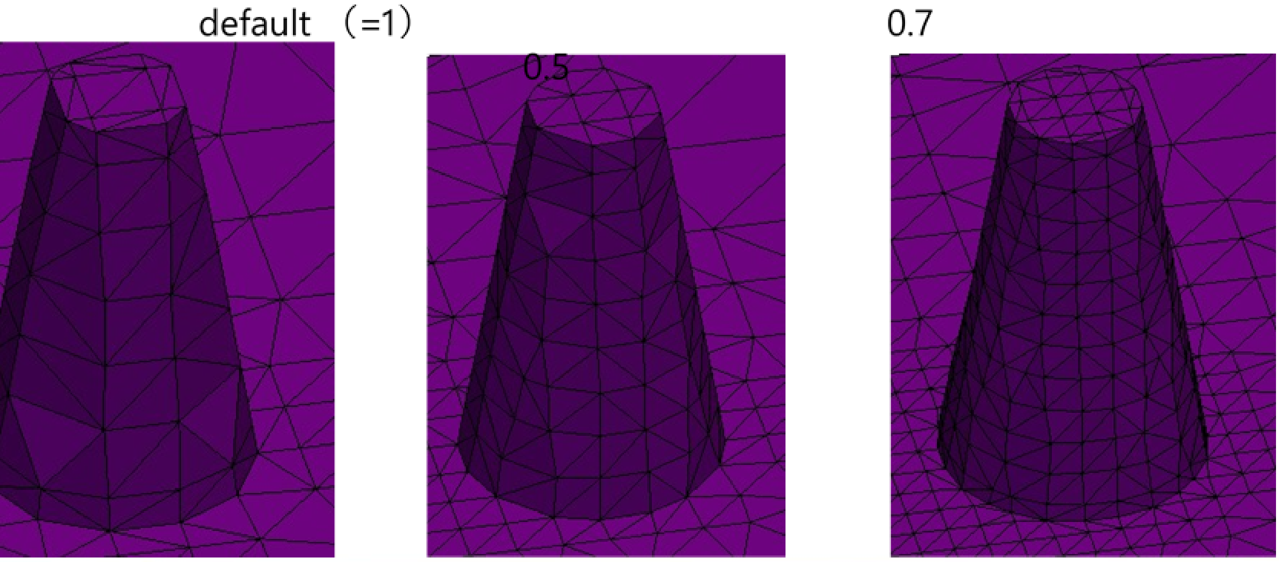
Explanation of Gap Close Refinement

- As Gap close tolerance is specified, this option is activated. The refinement and smoothing of the gap mesh will be carried out.



5． Settings of the wrapping mesh size (Contd)

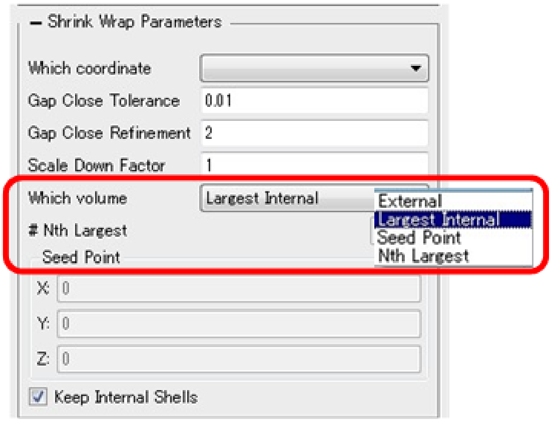
Explanation of Scale down factor
Run the wrapping with the value obtained by multiplying the set size with the ale factor.

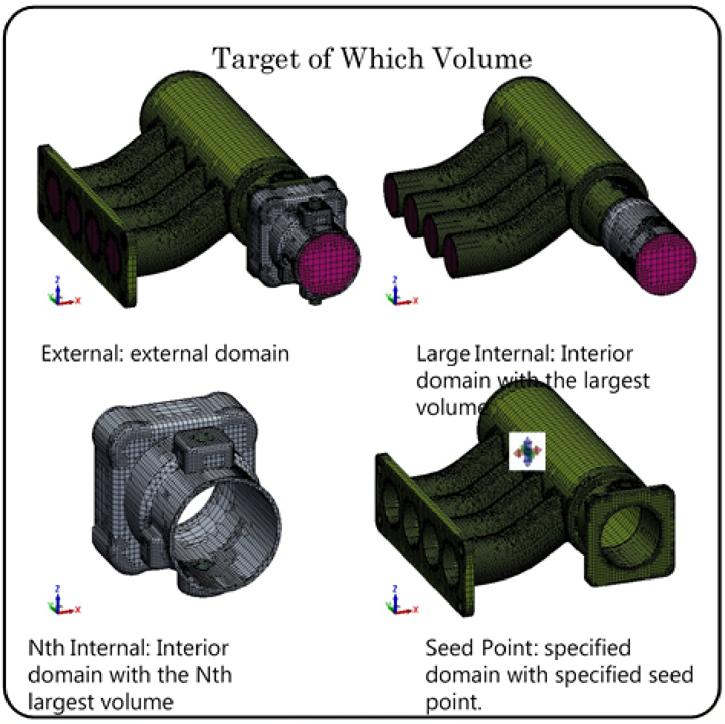


5． Settings of the wrapping mesh size (Contd)

Explanation of WhichVolume

- Select the target of wrapping. We will use Largest Internal this time

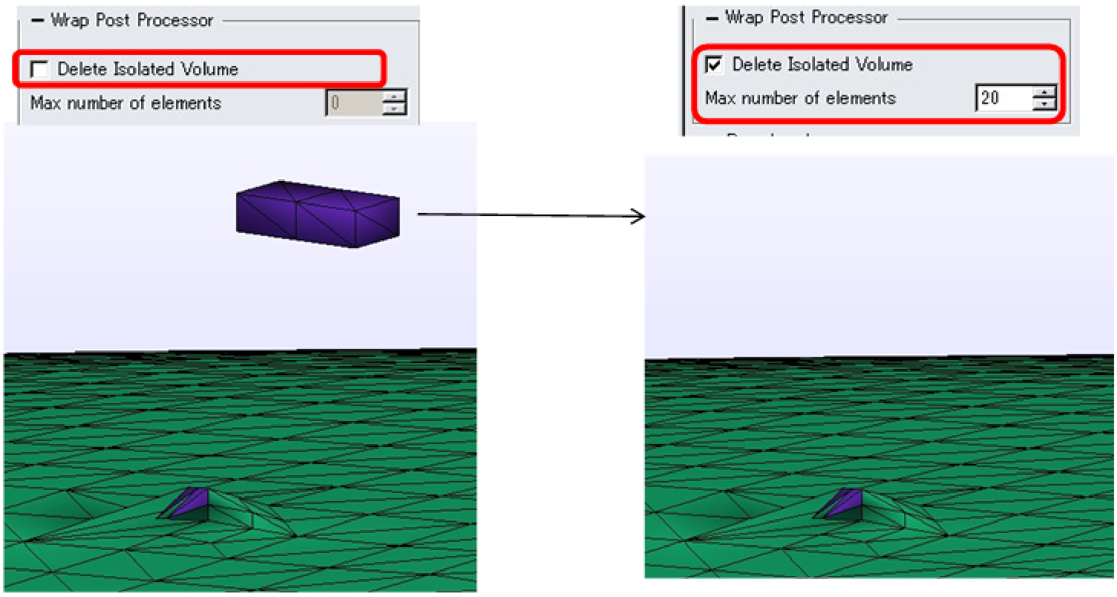




5． Settings of the wrapping mesh size (Contd)

Explanation of Delete Isolate volume

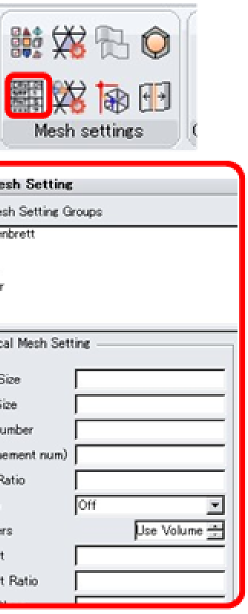
- Automatically delete the isolated volumes that are generated during wrapping.



5． Settings of the wrapping mesh size (Contd)

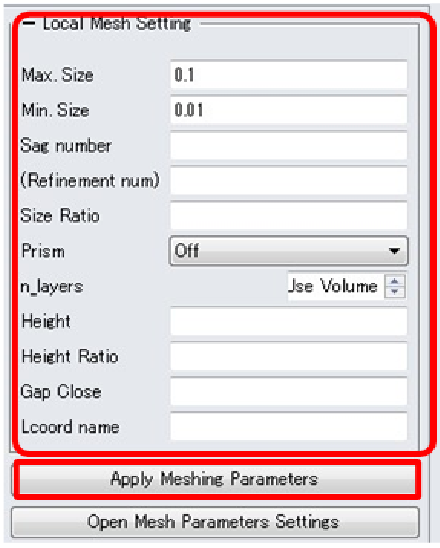
Settings of the local mesh 1/2)

- Mesh size could be modified for each group. First of all, set the local mesh size for the car model.



②：using shift and ctrl keys to select all parts except those started with Z_.

③：set as follows

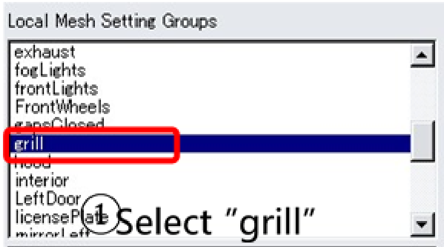


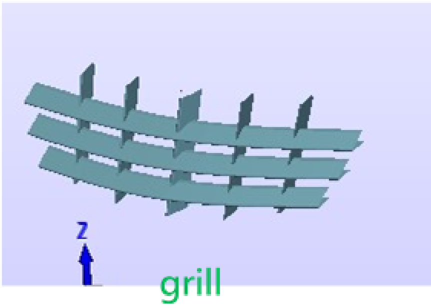
It not set, global values will be used.

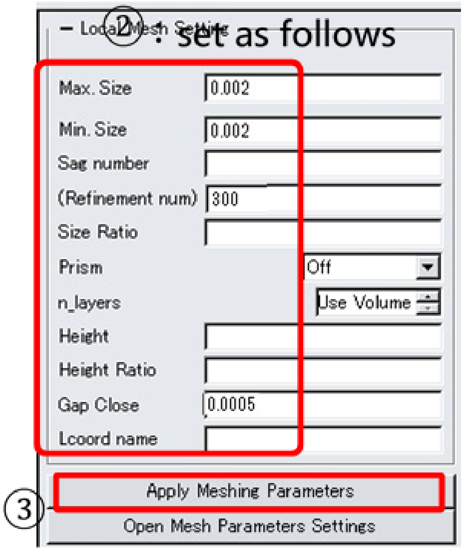
5． Settings of the wrapping mesh size (Contd)

Settings of the local mesh 2/2)

- Next, finer mesh will be set for the grill only.



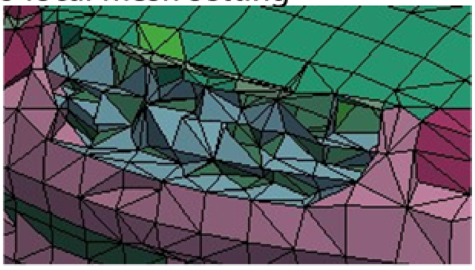




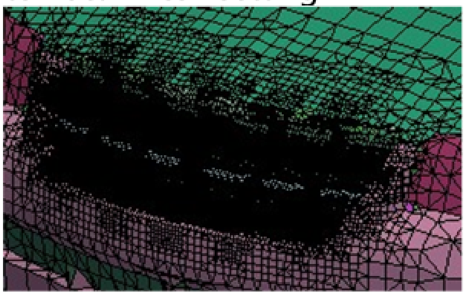
5． Settings of the wrapping mesh size (Contd)

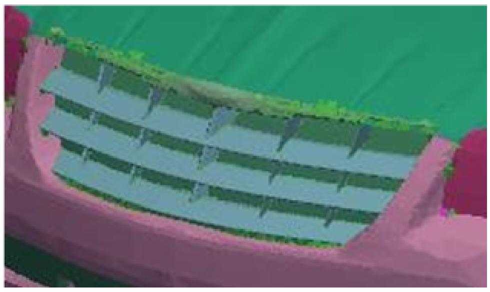
Visualization of the local mesh settings

No local mesh setting



After local mesh setting



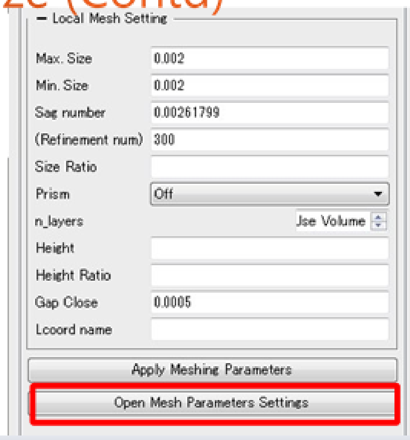


The shape is respected by the local mesh settings.

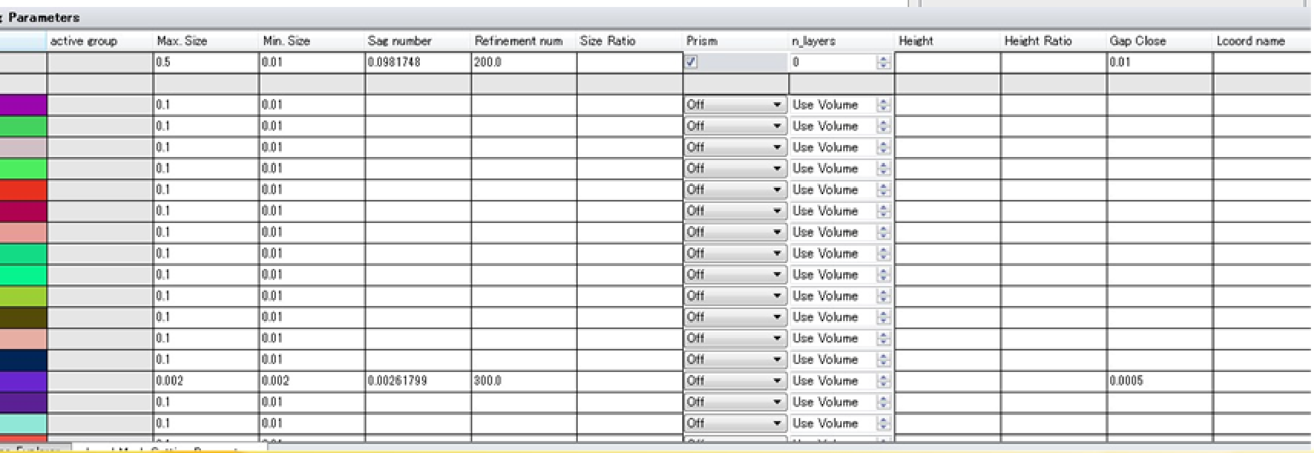
5．Settings of the wrapping mesh size (Contd)

Confirmation of the local mesh settings and the layers

- It is possible to check the local mesh size settings by the display list.
- It is also possible to change the settings by directly modifying in the spreadsheet.



①：open the spreadsheet.



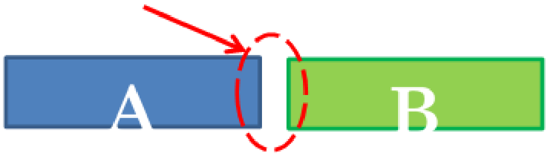
Parameters

	active group	Max. Size	Min. Size	Sag number	Refinement num	Size Ratio	Prism	n_layers	Height	Height Ratio	Gap Close	Lcoord name
		0.5	0.01	0.0981748	200.0		☑	0			0.01	
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				
		0.002	0.002	0.00261799	300.0		Off	Use Volume			0.0005	
		0.1	0.01				Off	Use Volume				
		0.1	0.01				Off	Use Volume				

6． Setting of the Thin cut

Thin cut is used for the following cases.

- In order to maintain the gap between A and B groups.

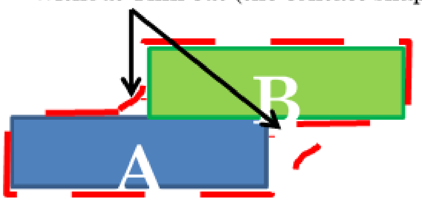


- In order to respect the contact shape between group A and B.

he following figures, the red dashed lines are the wrapping.

. Thin cut (the shape of the contact is respected.)

Without Thin cut (the contact shape is not
ected.)

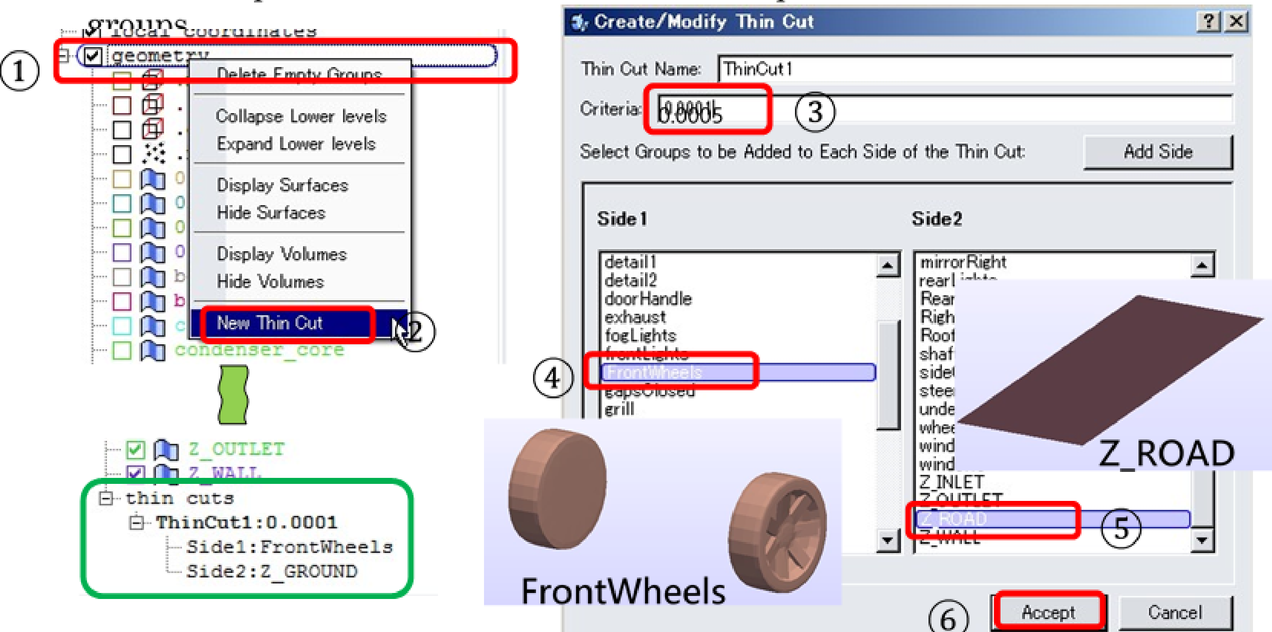


Regardless of the global or local mesh size, the mesh near the thin cuts will be refined to the specified criteria to respect the geometry.

6． Setting of the Thin cut (Contd)

Setting of Thin cut

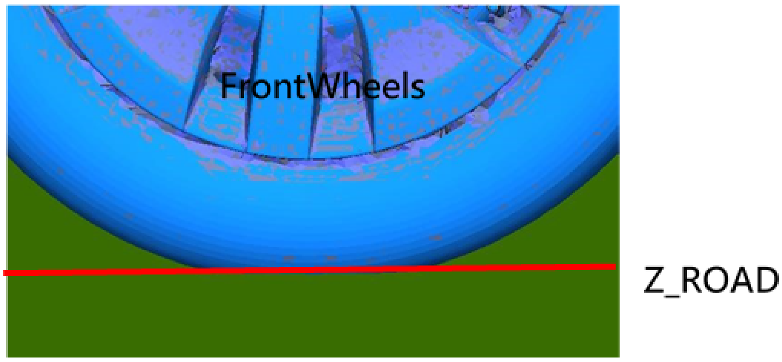
- In this manual, thin cut will be set for the frontWheels and the Z_ROAD in order to improve the resolution of the contact portion between these two groups.

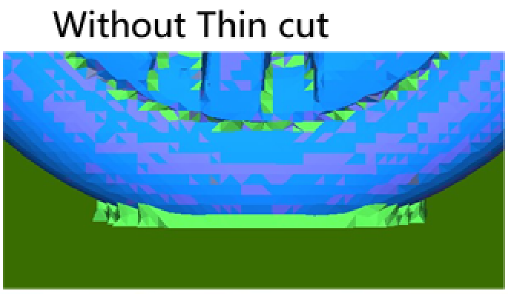


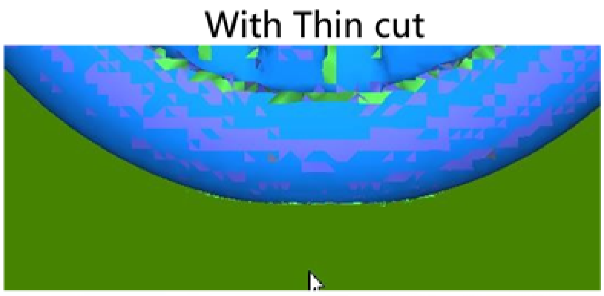
Thin cut is created in the geometry tree.

6． Setting of the Thin cut (Contd)

Effect of Thin cut



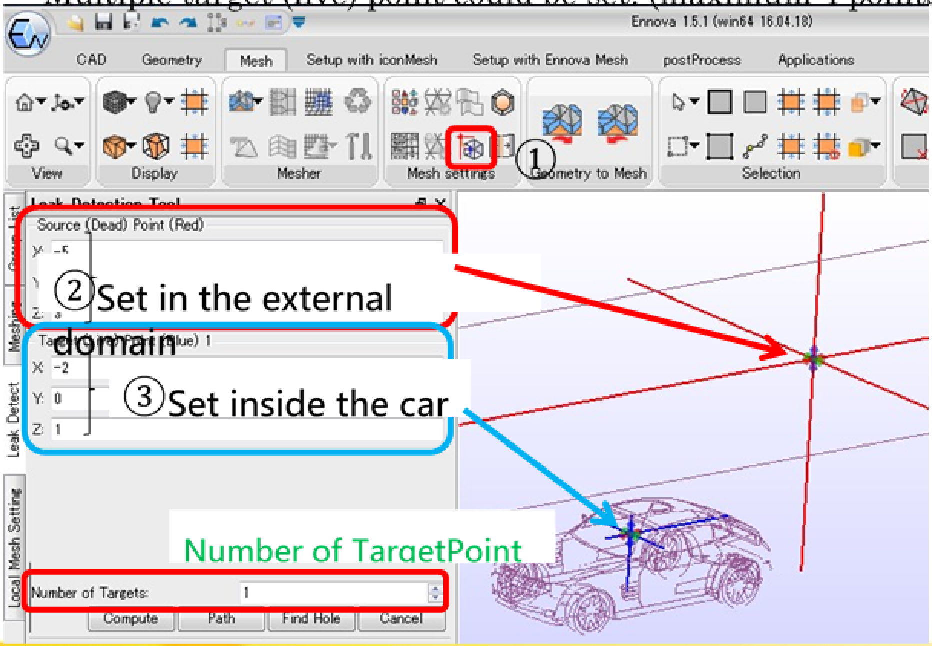




7． Setting the leak check

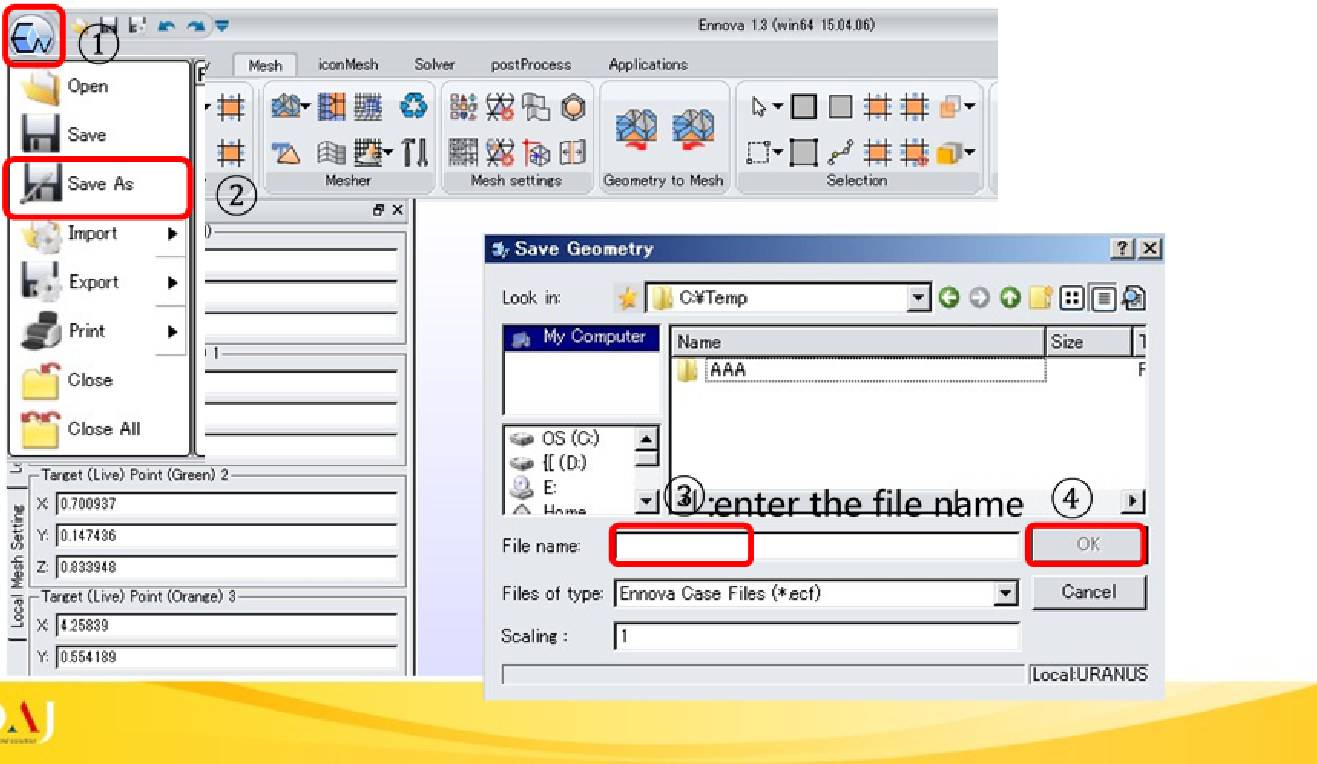
The leak check function is to check whether the wrapped surface has leak or not.

- By putting the source (dead) point in the external domain to the target (live) point, the leaks in the wrapped surface could be identified.
- Multiple target (live) point could be set. (maximum 4 points.)



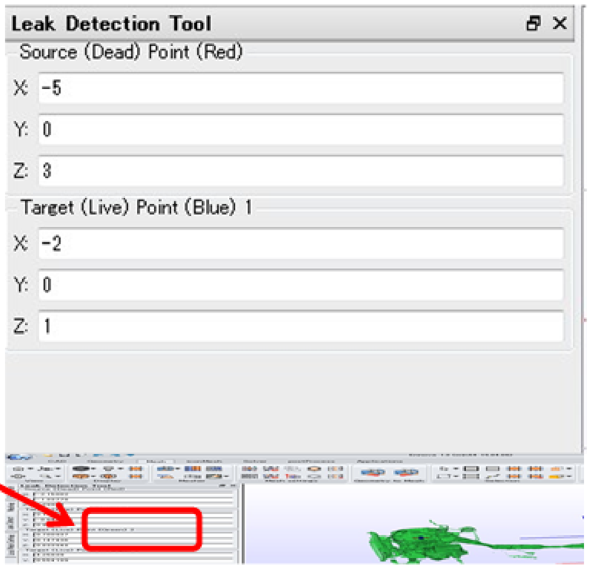
8. Saving the case data

Save the data (.ecf) before wrapping



9． Executing the surface wrapping and the results

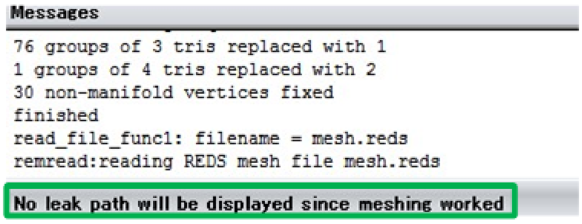
- Wrapping and leak check
  - Wrapping is executed together with the leak check.
  - It is also possible to run the wrapping only.

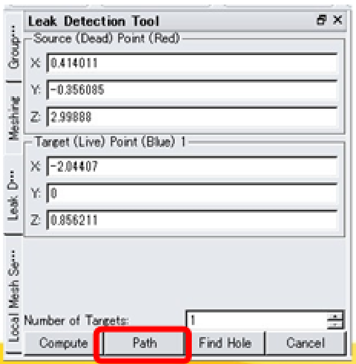


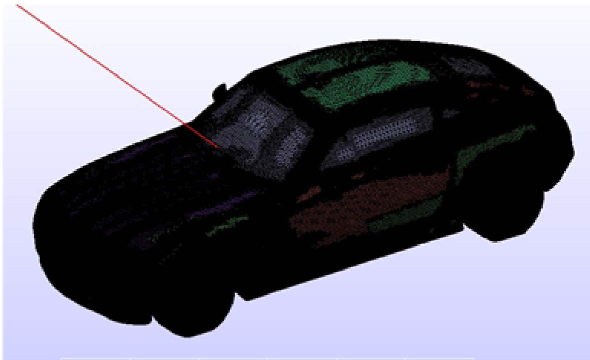
9. Executing the surface wrapping and the results (contd)

Leak check

- The information of leak check is displayed in the left bottom of GUI.
- When the black line is not displayed, leak does not exist.
- Click “Path” in the Leak Detection Tool to re-display the leak path.



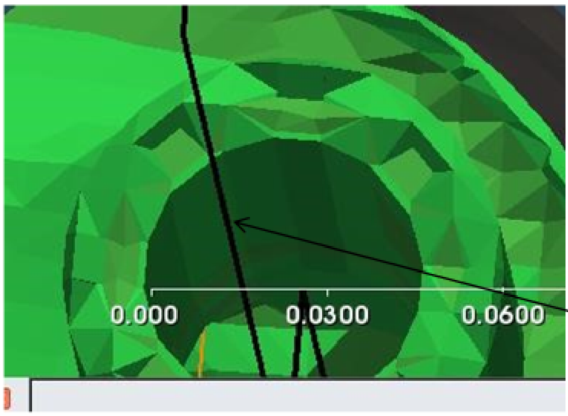


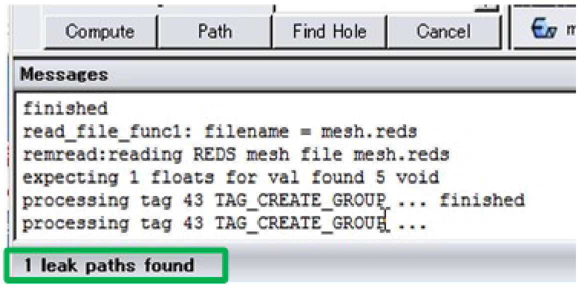


9． Executing the surface wrapping and the results (contd)

What to do if leak is detected(1/2)

- Leak exists as the message and the black line are displayed.
- For example, if a hole is larger than the gap close size, leak could occur.

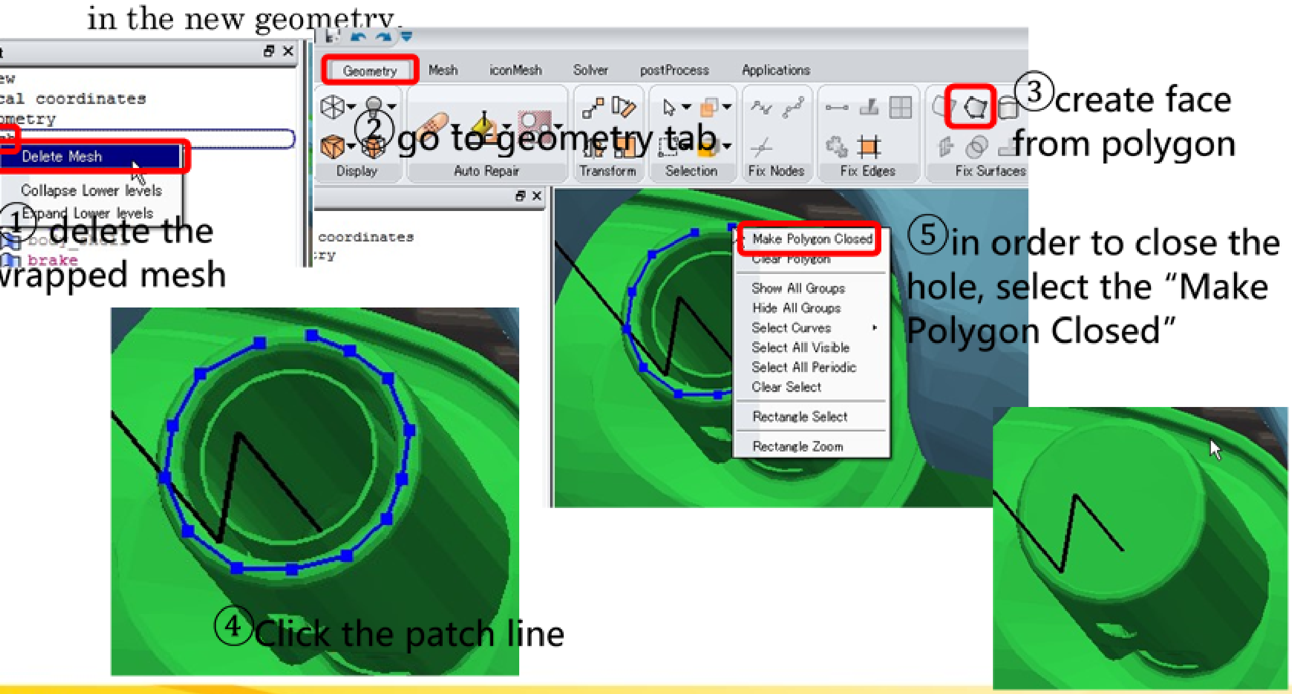




9． Executing the surface wrapping and the results (contd)

What to do if leak is displayed (2/2)

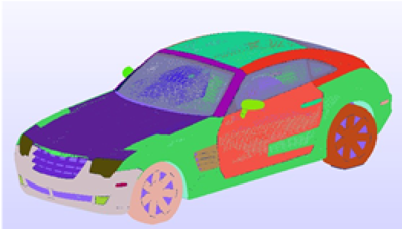
- Create a patch on the hole.
- After the hole is filled in, run leak check again to confirm that no leak exists in the new geometry.

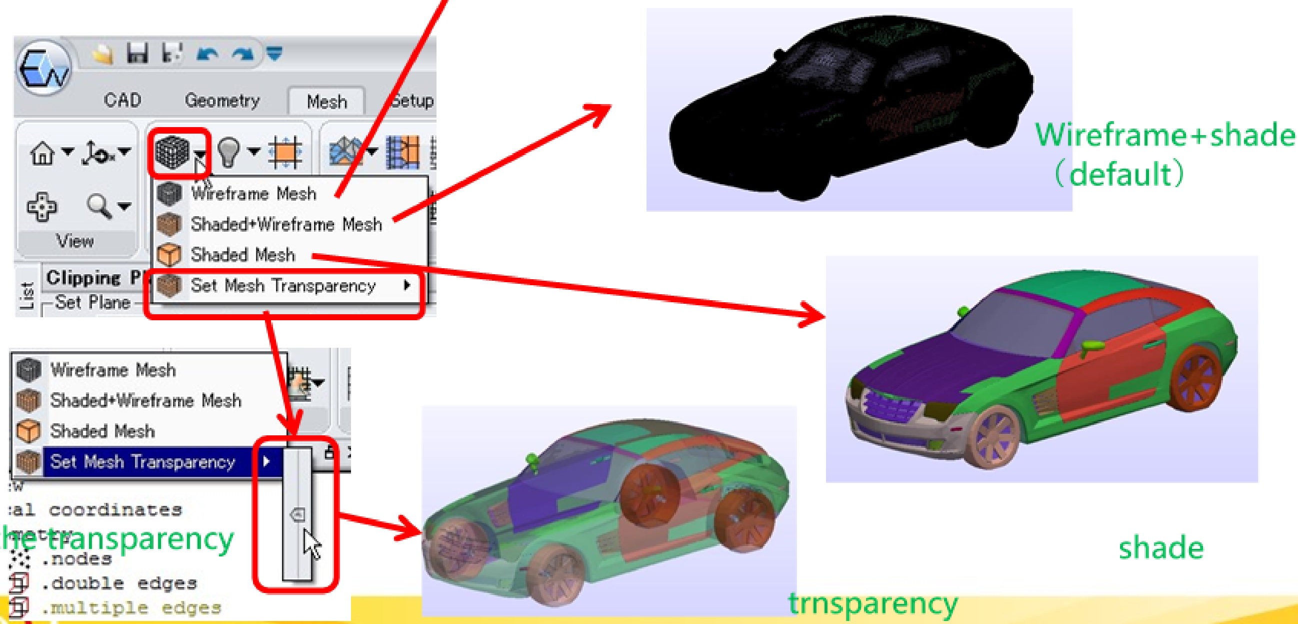


9. Executing the surface wrapping and the results (contd)

Changing the display method

- Here we introduce the methods of checking the generated mesh.

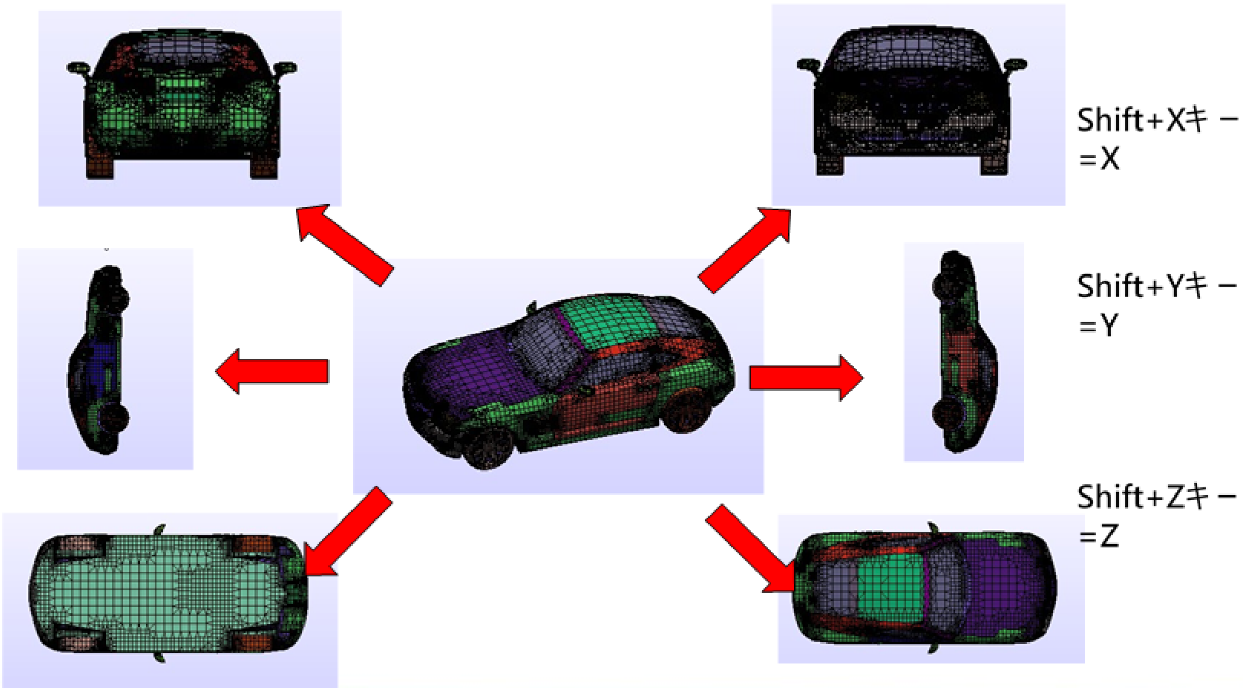




9． Executing the surface wrapping and the results (contd)

Change the viewpoint by shortkeys

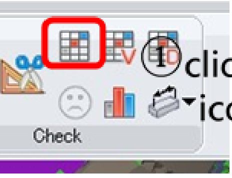
- X,Y,Z、 Shift+X 、 Shift+Y 、 Shift+Z



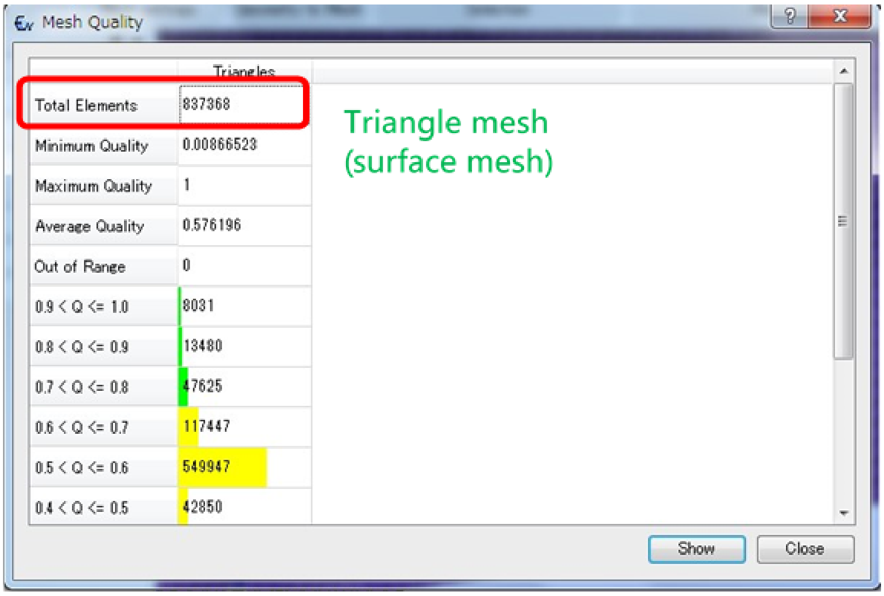
9． Executing the surface wrapping and the results (contd)

Confirm the mesh numbers

- Here introduces the methods to check the mesh numbers



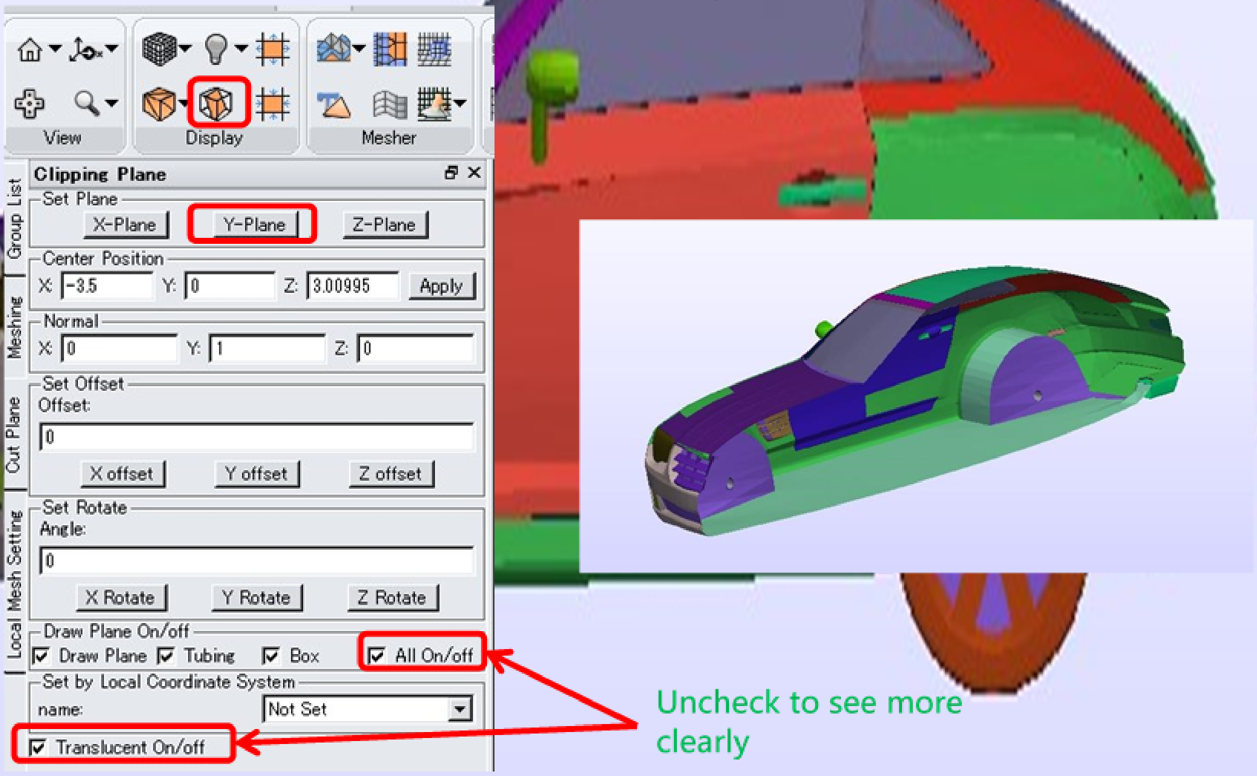
①click this icon



9. Executing the surface wrapping and the results (contd)

Display the clipping plane

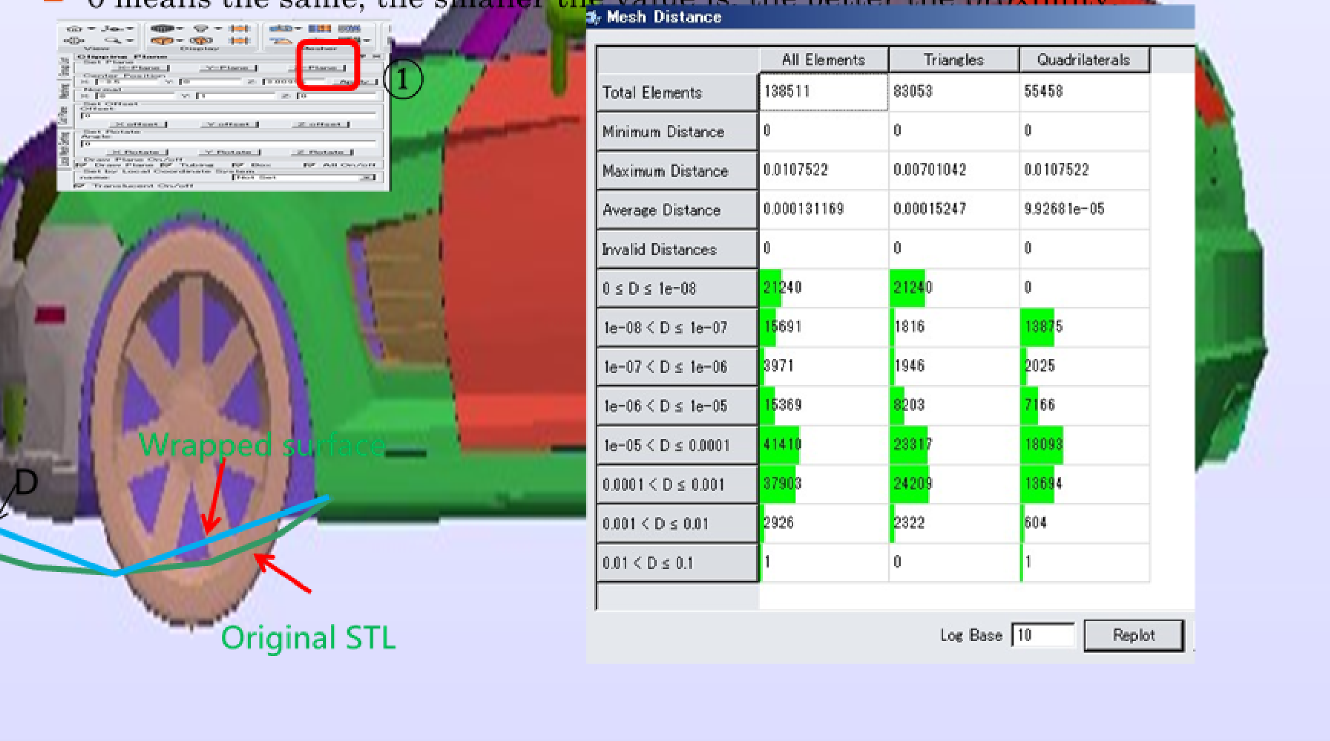
- Here we introduce how to display a clipping plane



9． Executing the surface wrapping and the results (contd)

Checking the shape difference between the Geometry (original STL) and the wrapped surface

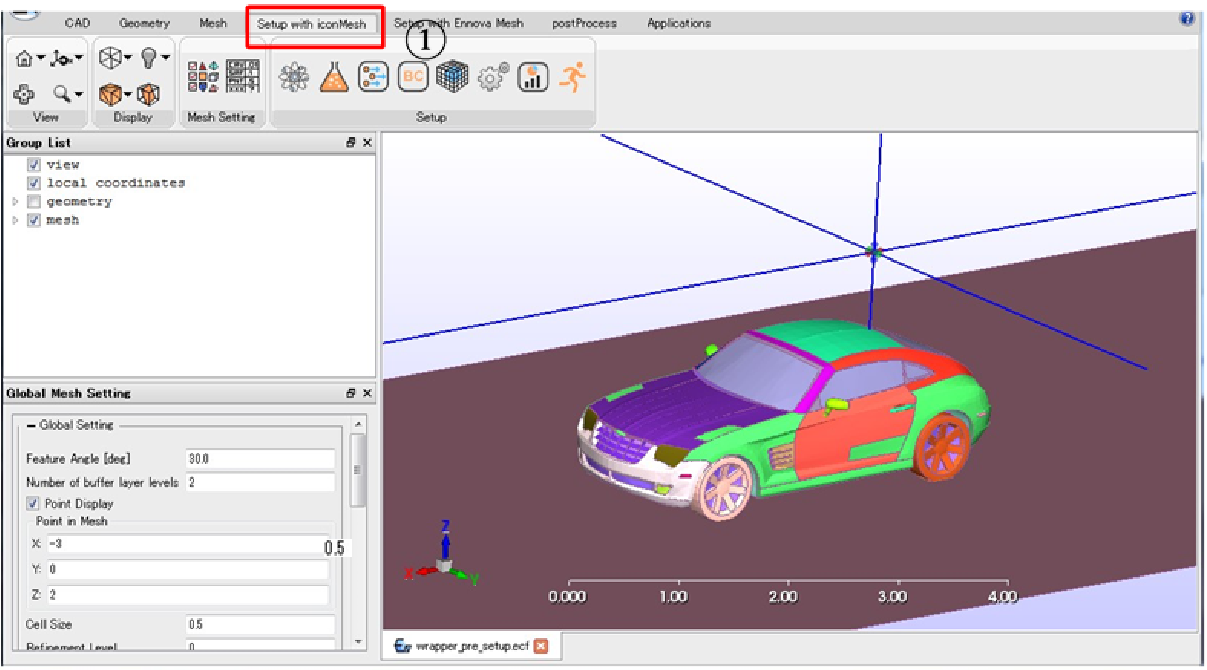
- Checking the proximity between the geometry and the wrapped surface.
- 0 means the same, the smaller the value is, the better the proximity.



	All Elements	Triangles	Quadrilaterals
Total Elements	138511	83053	55458
Minimum Distance	0	0	0
Maximum Distance	0.0107522	0.00701042	0.0107522
Average Distance	0.000131169	0.00015247	9.92681e-05
Invalid Distances	0	0	0
0 ≤ D ≤ 1e-08	21240	21240	0
1e-08 < D ≤ 1e-07	15691	1816	13875
1e-07 < D ≤ 1e-06	3971	1946	2025
1e-06 < D ≤ 1e-05	15369	8203	7166
1e-05 < D ≤ 0.0001	41410	23317	18093
0.0001 < D ≤ 0.001	37903	24209	13694
0.001 < D ≤ 0.01	2926	2322	604
0.01 < D ≤ 0.1	1	0	1

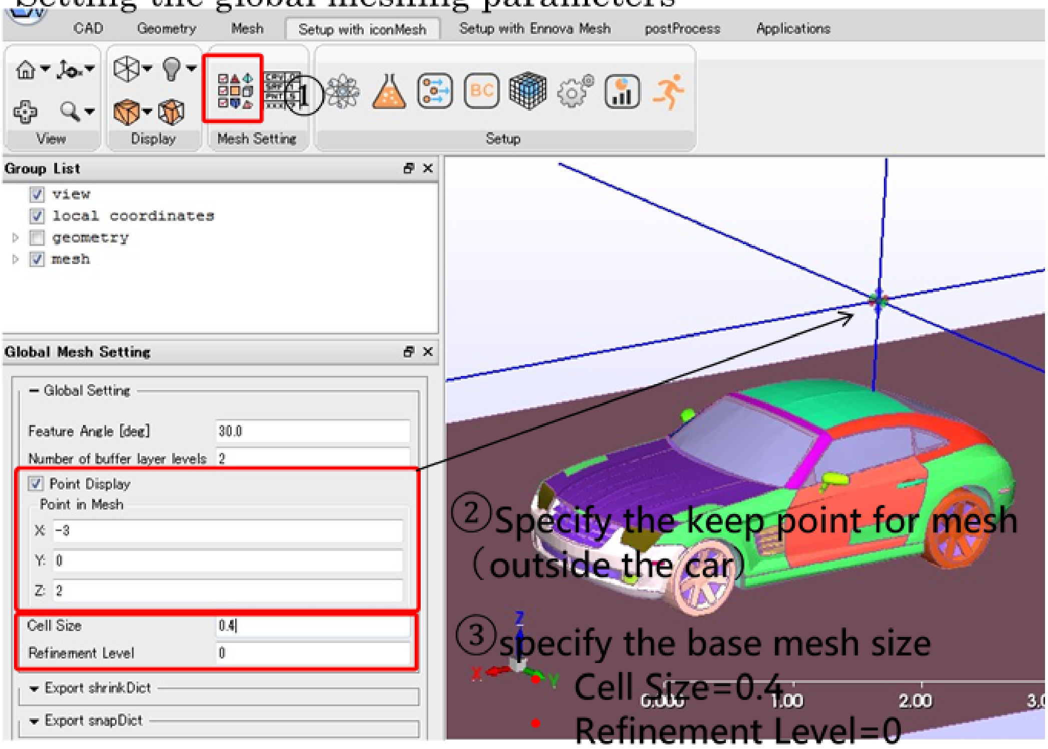
１０．Setting the iconCFD mesher

Use iconCFD mesher to generate the volume mesh. The settings will be carried out in the Setup with iconMesh.



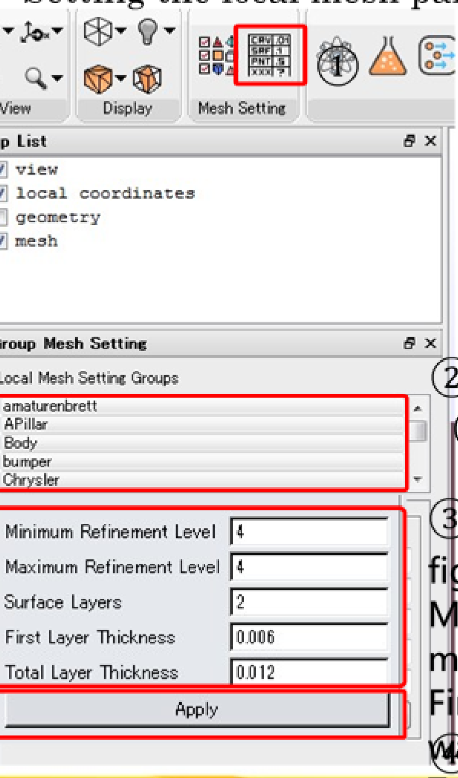
１０． Setting the iconCFD mesher (Contd)

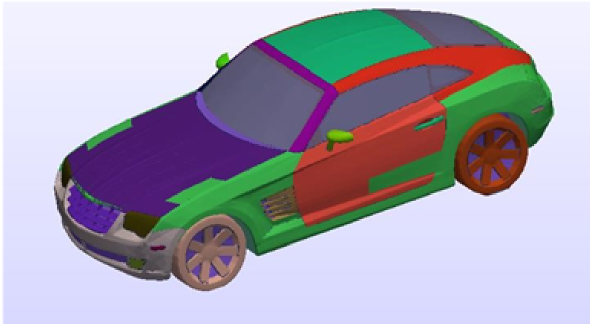
Setting the global meshing parameters



１０．Setting the iconCFD mesher (Contd)

Setting the local mesh parameters (1/3)





②select all the geometry groups on the car (by ctrl and shift keys)

③set the local mesh parameter as shown in the left figure.

Max/MinRefinement: refinement levels from the base mesh

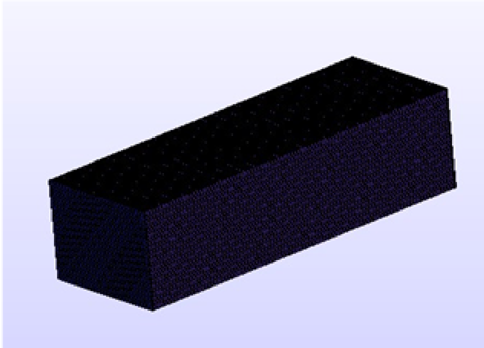
FirstLayerThickness: the thickness of the first layer from wall

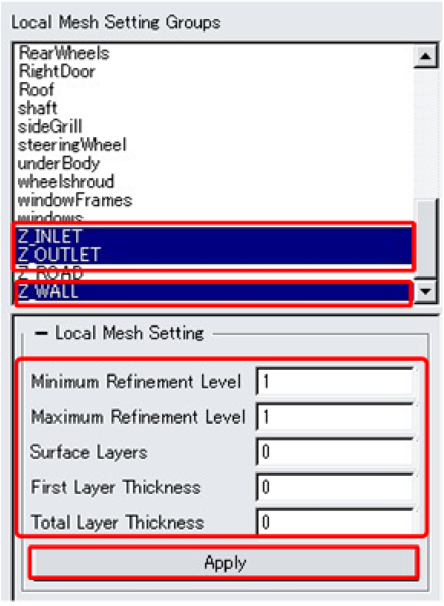
TotalLayerThickness: total thickness of the layers

④click “Apply”

１０． Setting the iconCFD mesher (Contd)

Setting the local mesh parameters(2/3)





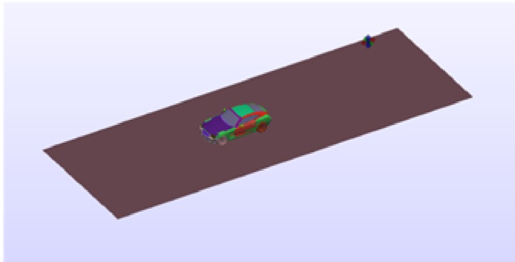
①Select Z_INLET,　Z_OUTLET,　Z_WALL

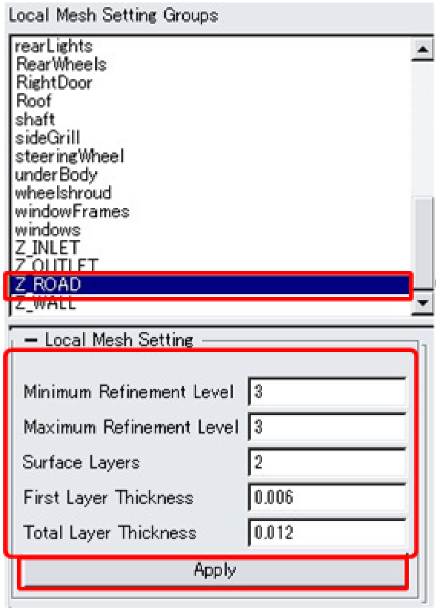
②Set the local mesh parameter
Surface Layers is set to 0 since layers are not needed near the wall.

③Click "Apply"

１０．Setting the iconCFD mesher (Contd)

Setting the local mesh parameters (3/3)





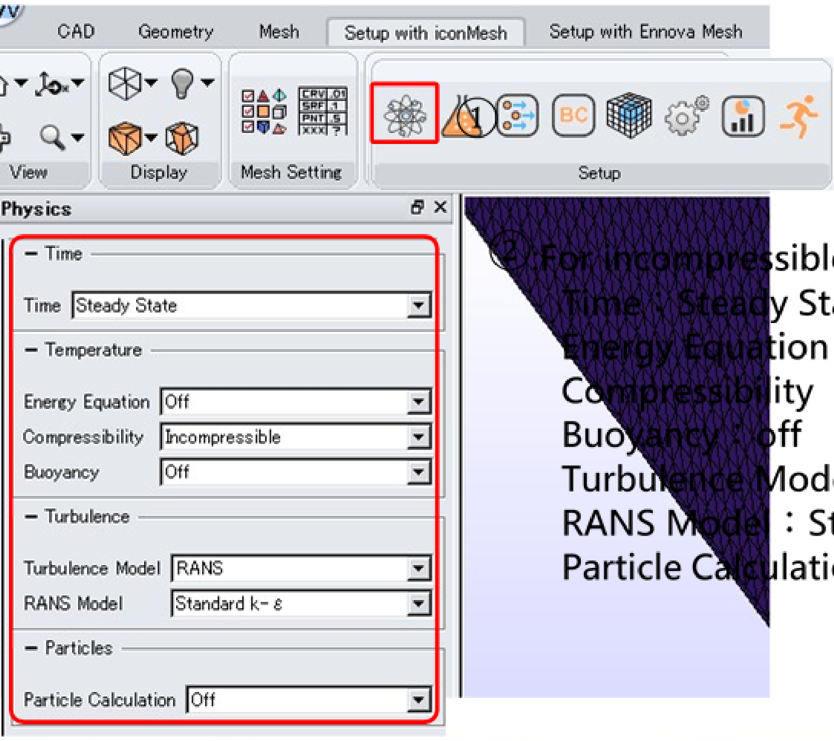
①Select "Z_ROAD"

②Set the local mesh parameters as shown in the left figure

③Click "Apply"

１１. Setting the iconCFD solver

Setting the Physics

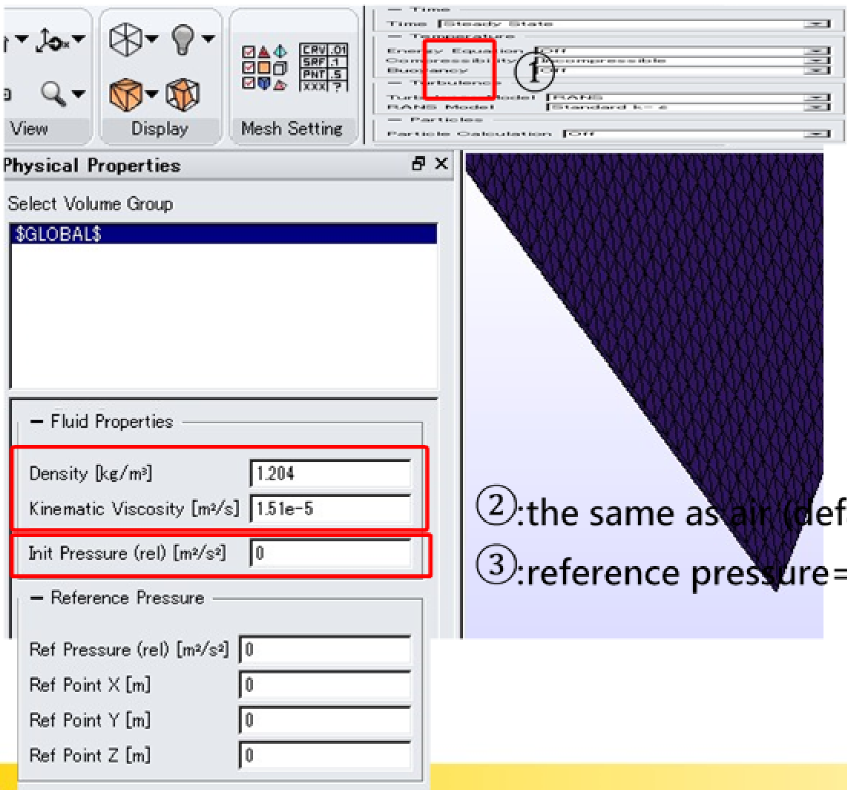


②For incompressible steady state simulations:

Time：Steady State
Energy Equation：off
Compressibility ：Imcompressible
Buoyancy：off
Turbulence Model：RANS
RANS Model ：Standard k-ε
Particle Calculation：off

1 1 . Setting the iconCFD solver (Contd)

Physical Properties



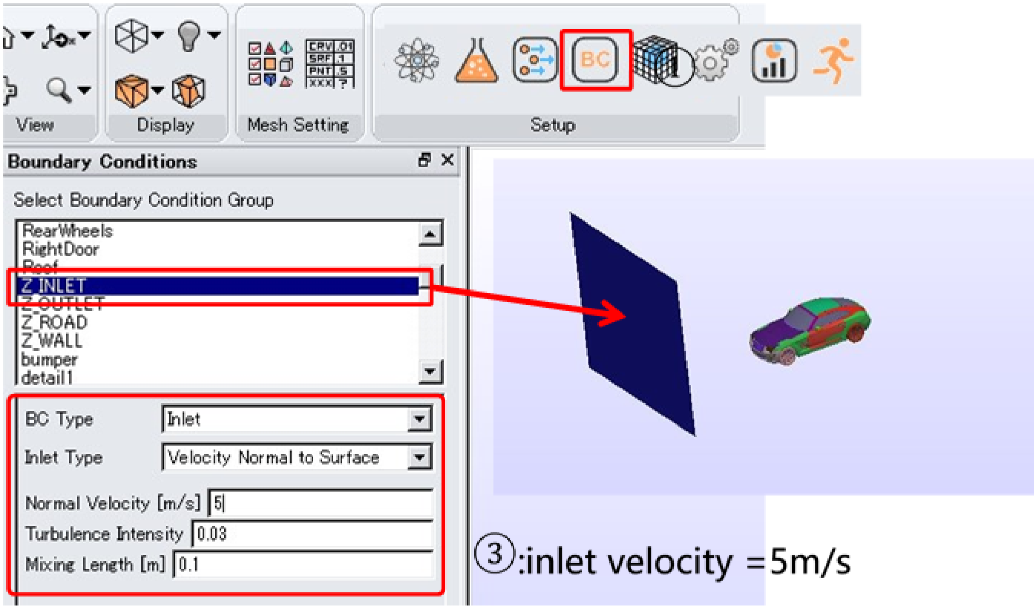
②:the same as air (default)

③:reference pressure=0 (default)

１１. Setting the iconCFD solver (Contd)

Boundary condition (inlet)

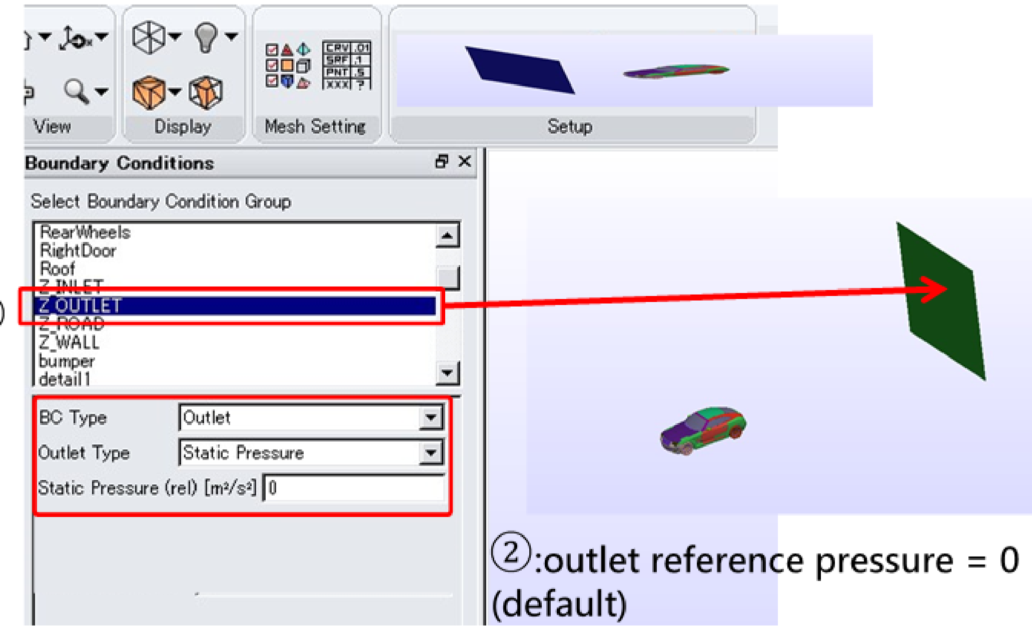
- velocity.



11. Setting the iconCFD solver (Contd)

Boundary condition (outlet)

- pressure

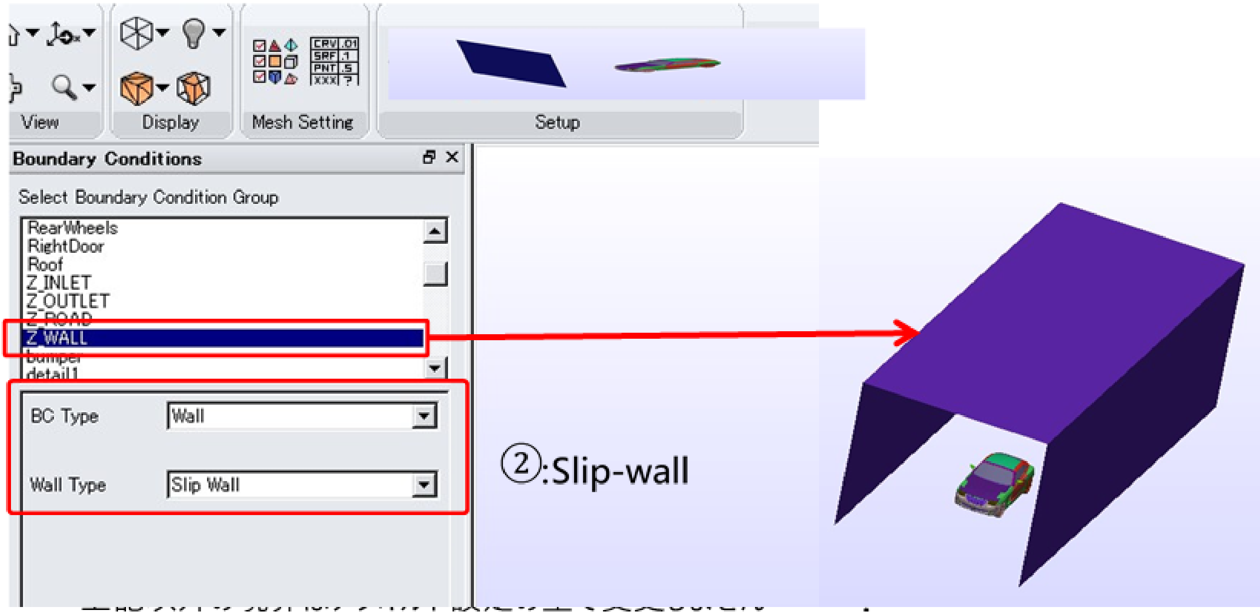


②:outlet reference pressure = 0 (default)

1 1 . Setting the iconCFD solver (Contd)

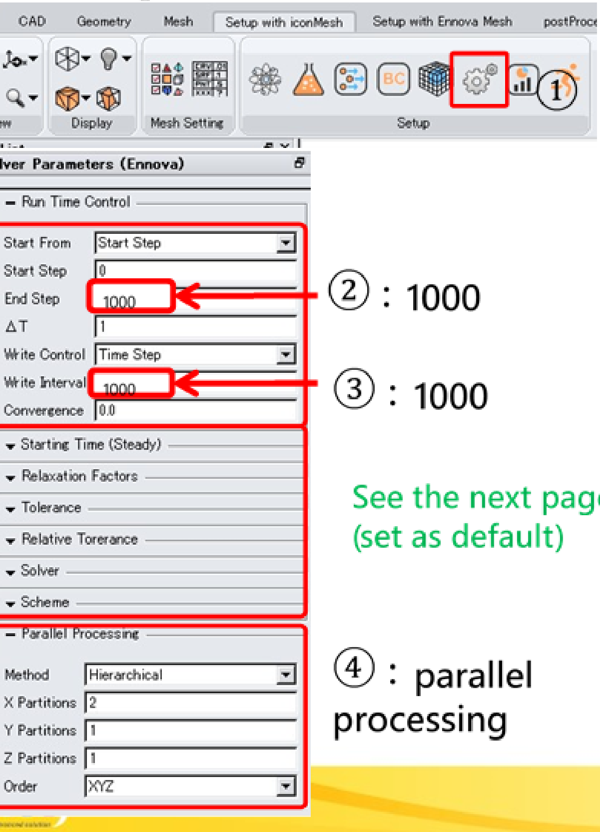
Boundary condition (boundary walls)

- Slip wall



１１. Setting the iconCFD solver (Contd)

Solver parameter



② : 1000

③ : 1000

See the next pages (set as default)

In this manual, except 2 and 4, the others are set as default.

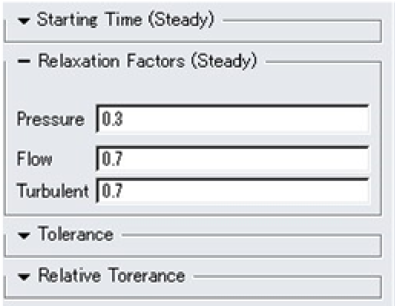
④ : parallel processing

1 1 . Setting the iconCFD solver (Contd)

Solver parameter

- Relaxation factors
- Use the default value in this manual

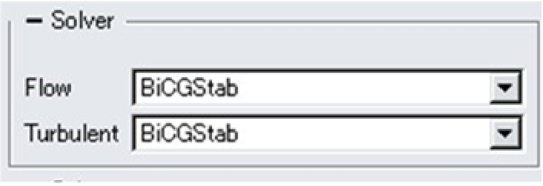
A) pressure
B) flow
C) turbulence



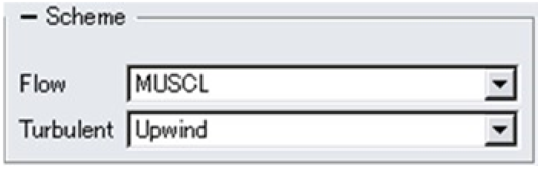
1 1 . Setting the iconCFD solver (Contd)

Solver parameter

- Algebraic solver
- Use the defaults in this manual



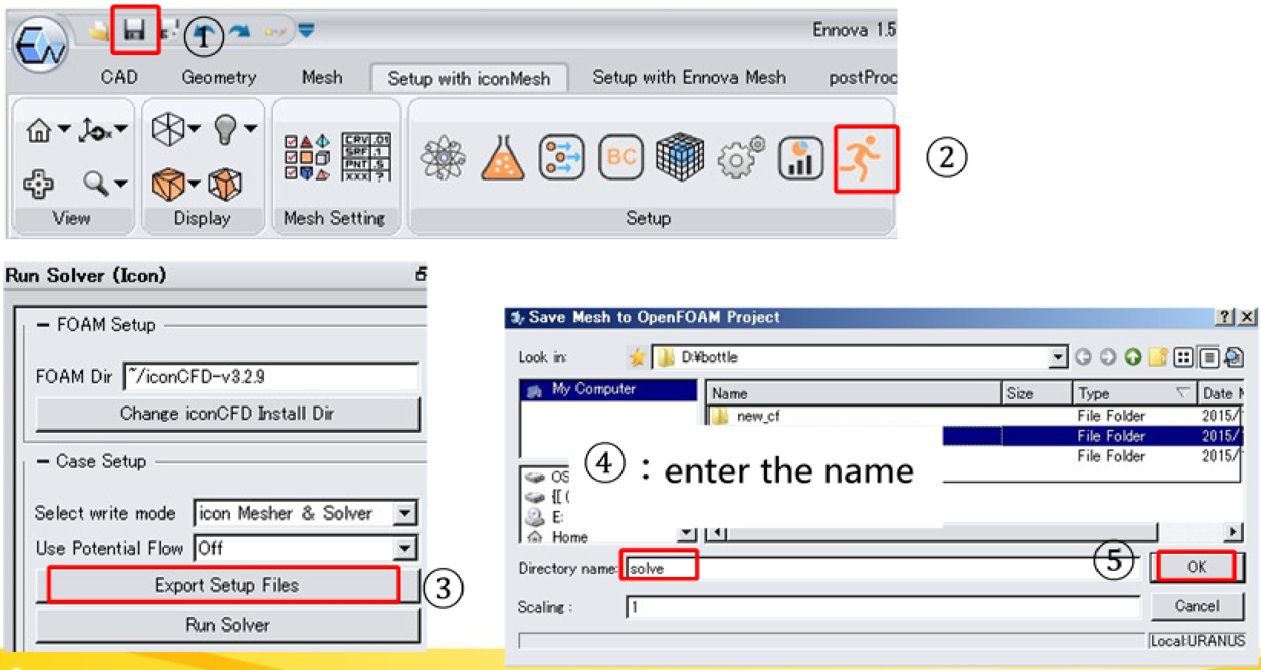
- Differencing schemes
- Use the defaults in this manual



1 1 . Setting the iconCFD solver (Contd)

Output of the iconCFD setting files

- Save the .ecf file for further use in iconCFD



１２．Running the simulation

After the mesh and solver settings in the Windows terminal, copy the files to a system where iconCFD could run (such as simulation server). Execute runHPC.sh to start simulations.

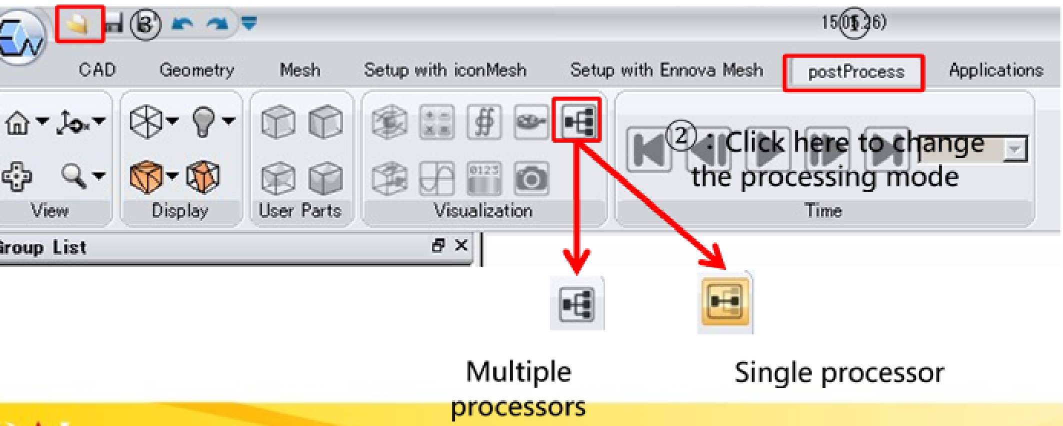
1 3. PostProcessing

Reading the simulation data from iconCFD (1/3)

- In the case of multiple processors
  - Put ***system*, *constant*, *processor**, ***.foam** files to a location accessible by Windows-PC.
- In the case of single processor
  - Put ***system*, *constant*, *time directories*, ***.foam files** to a location accessible by Windows-PC.

Selecting the postProcessing method

- Before reading the files, choose to process the results using multiple processors or single processor.

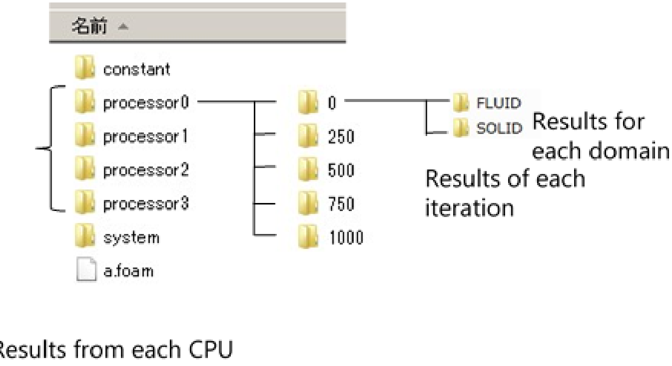


1 3. PostProcessing (Contd)

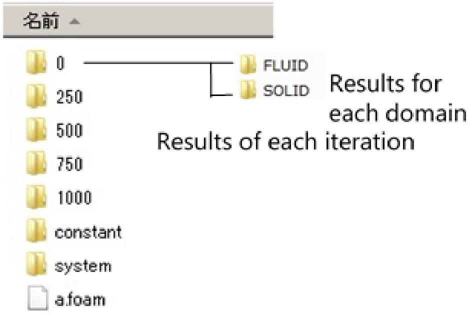
Reading the simulation data from iconCFD (2/3)

According to whether multiple or single processor(s) are(is)employed, these files are generated by the specified frequency (set in the Solver Parameter, p. 46).

- In the case of multiple processors



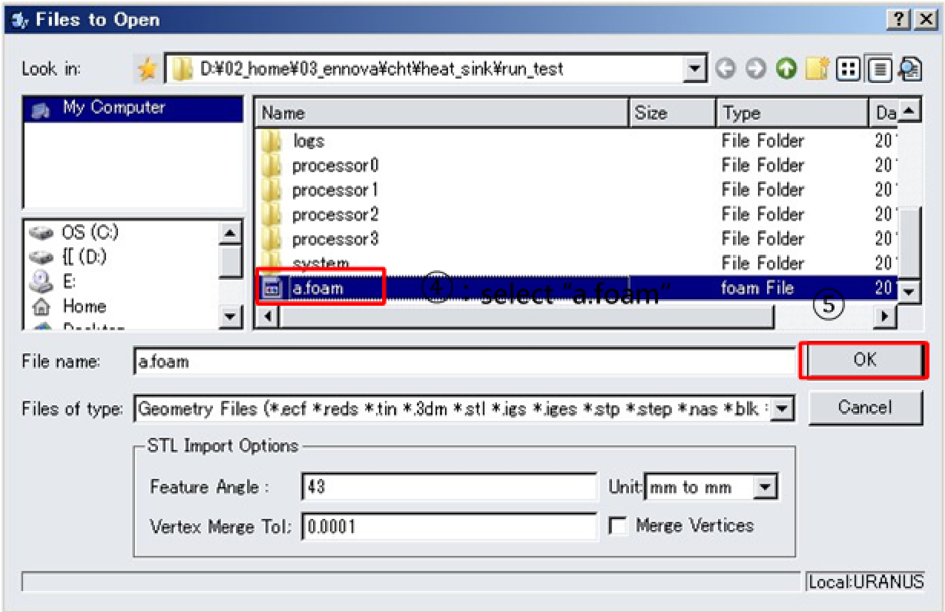
- In the case of single processor



1 3. PostProcessing (Contd)

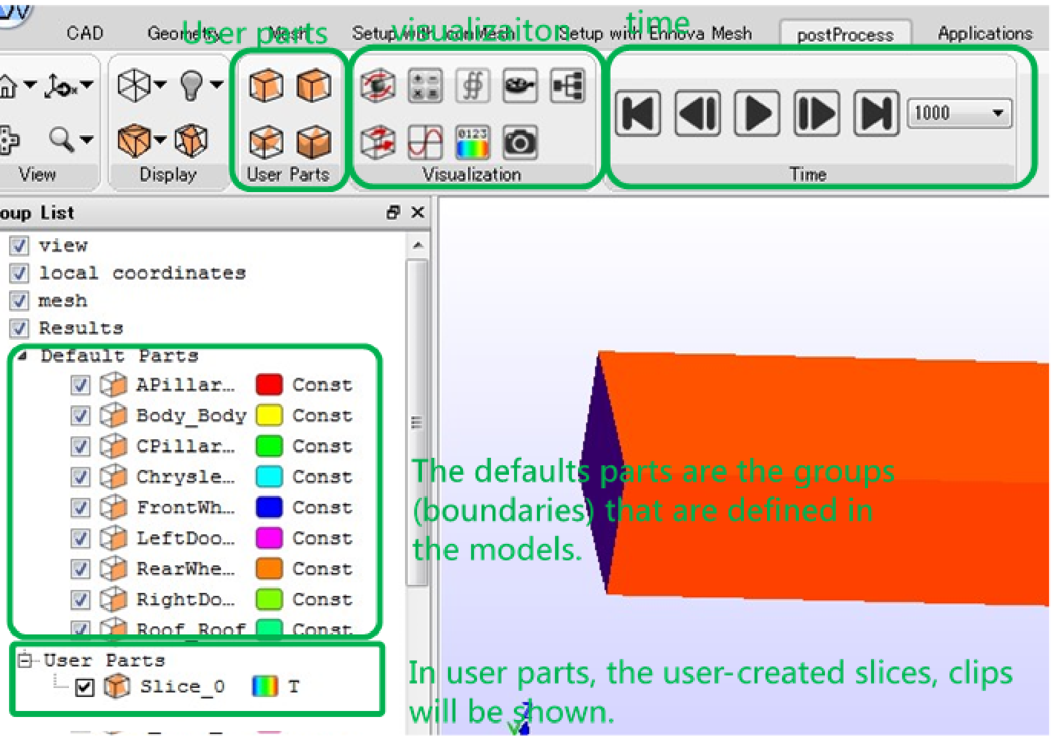
Reading the simulation data from iconCFD (3/3)

- The simulation data could be read via the file "a.foam" in the folder. "a.foam" is automatically generated using runHPC.sh.



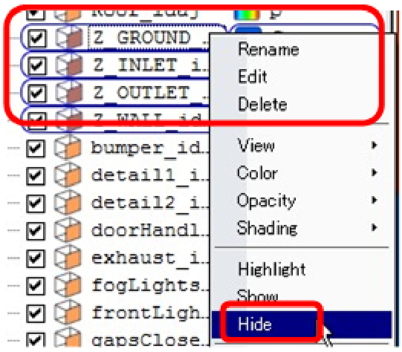
13. PostProcessing (Contd)

Postprocessing GUI



１３． Postprocessing (Contd)

Display and hide the groups



Hide the groups: select the outer boundaries, right click and select Hide, or select Show to display.

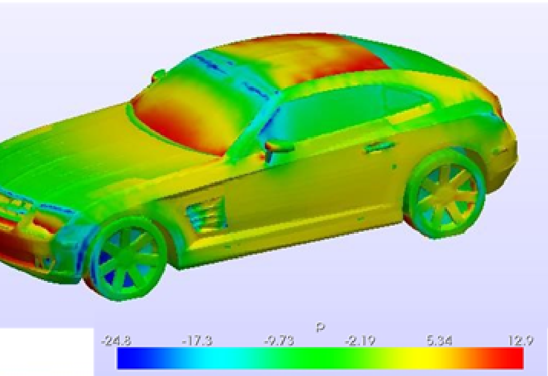
Display only the car model

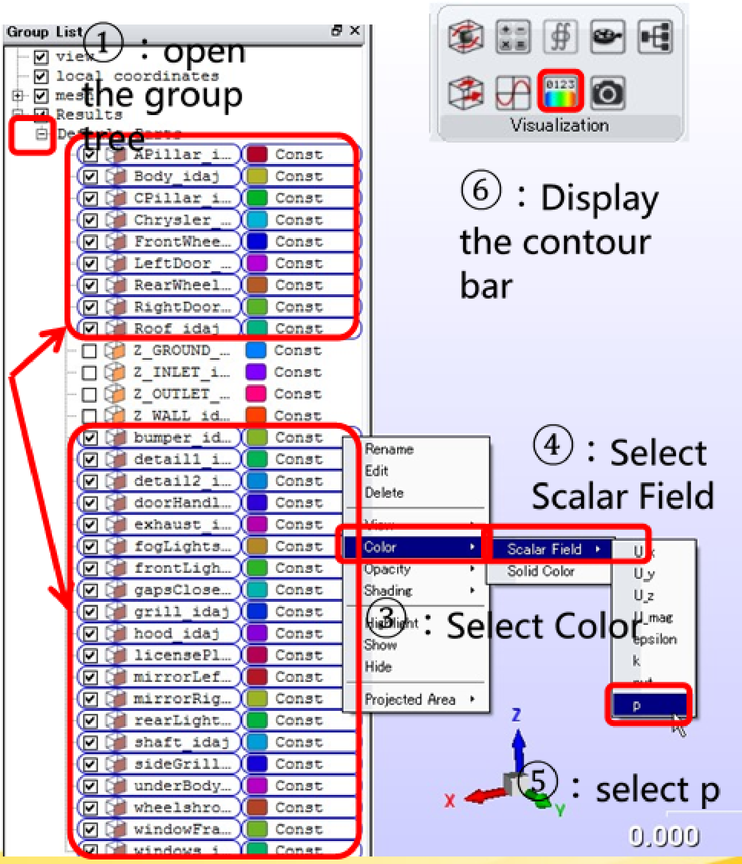
１３． Postprocessing (Contd)

Display scalars in groups

- The variables on the wall could be displayed on the surfaces of the groups.
- Select the car surface group and display pressure.

② : Select the groups that needs to be displayed.

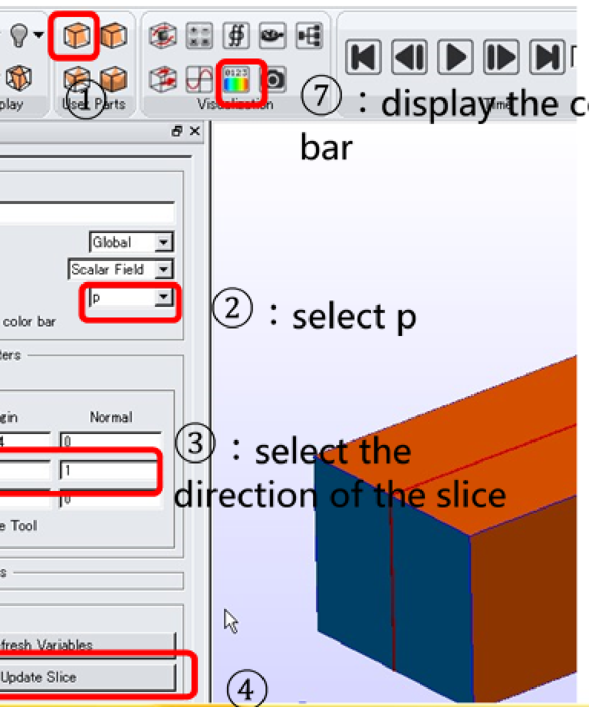




13. Postprocessing (Cont'd)

Pressure distribution on the slice

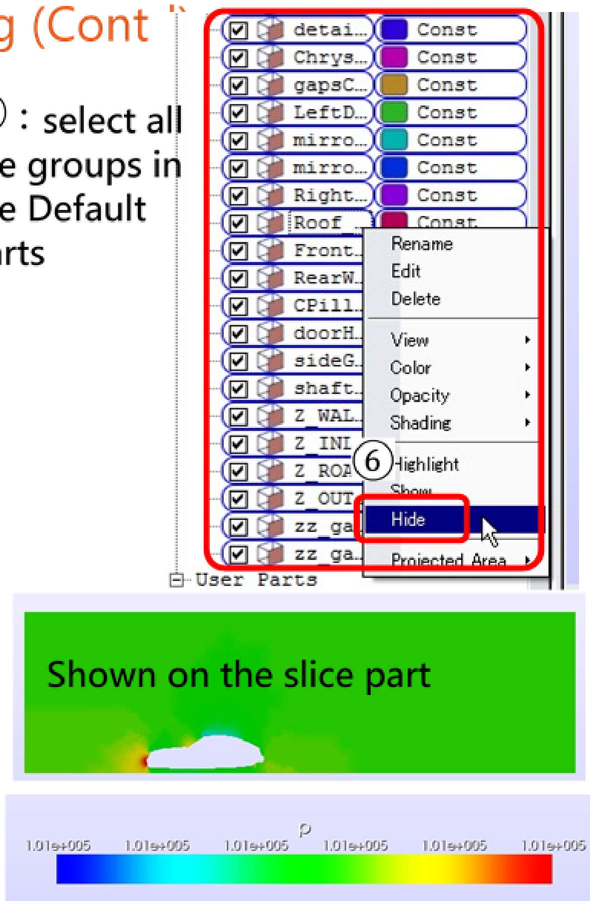
⑤：select all the groups in the Default Parts



⑦：display the contour bar

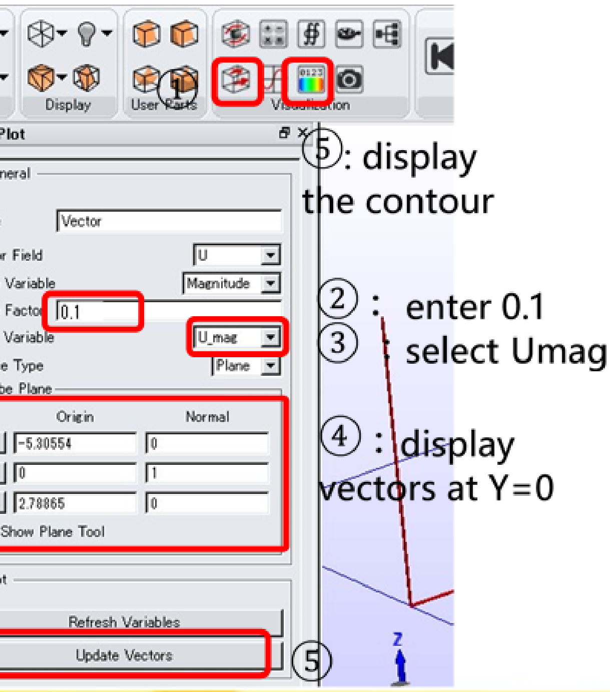
②：select p

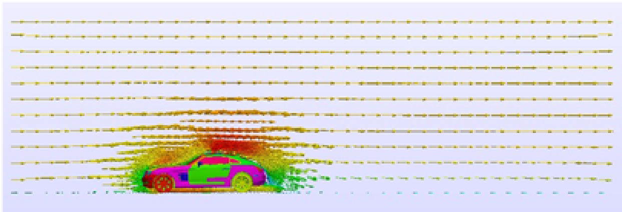
③：select the direction of the slice



１３． Postprocessing (Contd)

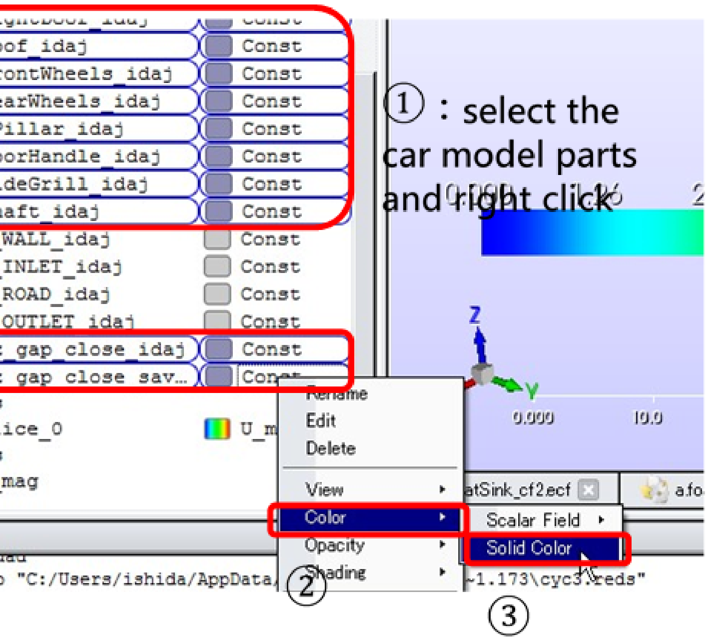
Display the velocity vector

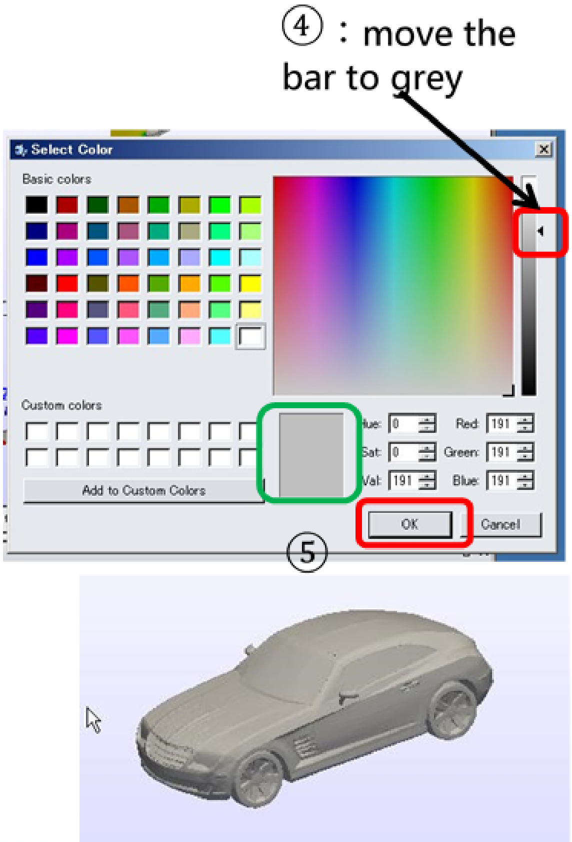


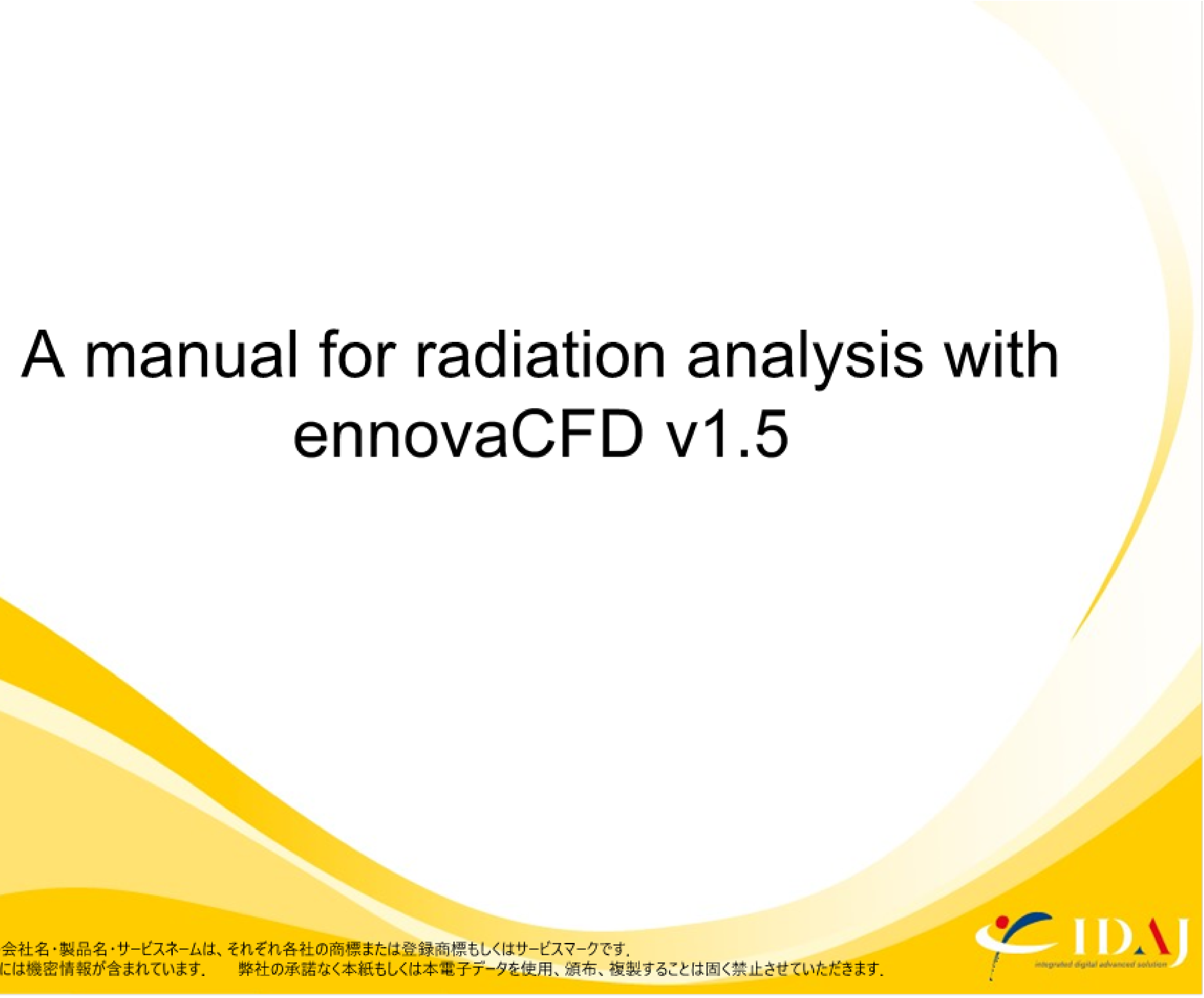


１３．Postprocessing (Contd)

Display the car model with single color






A manual for radiation analysis with
ennovaCFD v1.5
会社名・製品名・サービスネームは、それぞれ各社の商標または登録商標もしくはサービスマークです.
には機密情報が含まれています.　弊社の承諾なく本紙もしくは本電子データを使用、頒布、複製することは固く禁止させていただきます.
IDAJ
integrated digital advanced solution

Contents

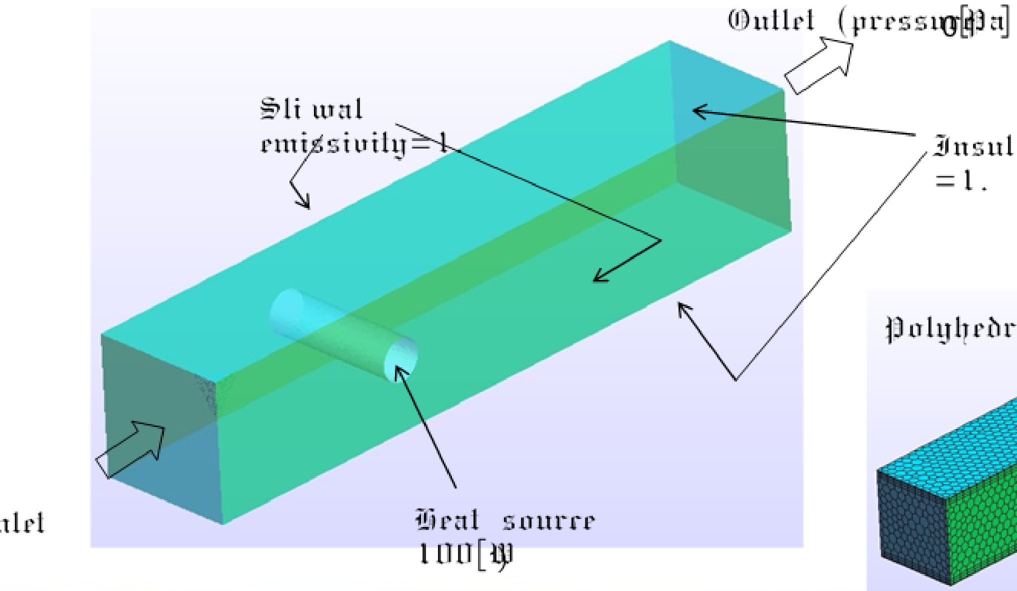
argeted version:
ennovaCFD1.5
iconCFD3.2.11

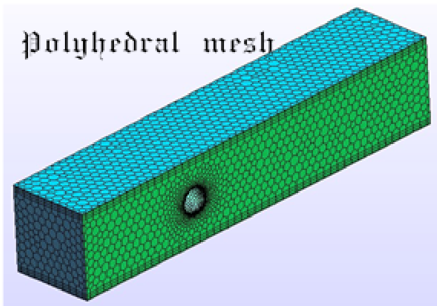
1. Settings for the thermal radiation analysis

1-1. Introduction

We setup up the analysis of thermal radiation (S2S model) with ennovaCFD v1.5 and iconCFD3.2.11 in this manual.

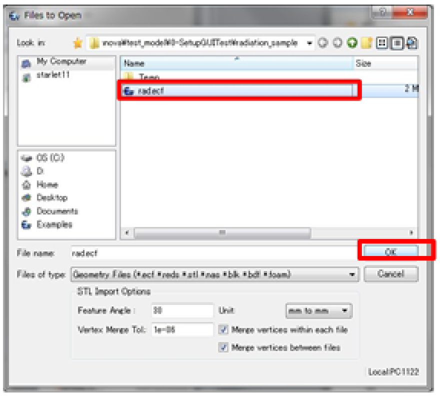
The rad.ecf file in the sample data will be used. It is the meshed data (polyhedral mesh) generated by ennovaCFD.



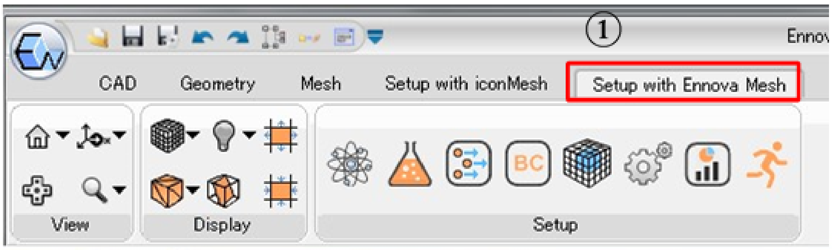


1-2 Settings of the iconCFD solver

Import the rad.ecf file



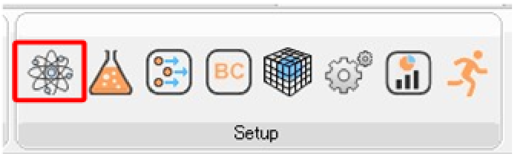
Since polyhedral mesh has already been generated by ennovaCFD, we proceed to Setup with Ennova Mesh.

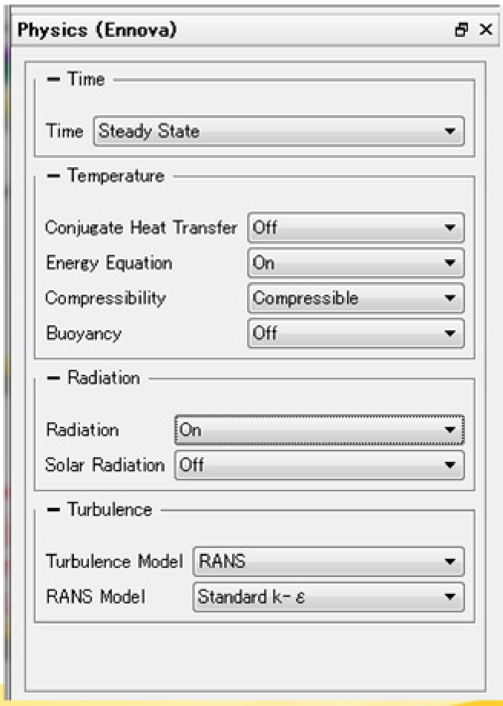


1-2 Settings of the iconCFD solver

In the Physics panel, steady/unsteady, compressible/incompressible, temperature on/off, turbulence model, etc., will be specified. In the current case, steady state, compressible, radiations will be set as the right figure.

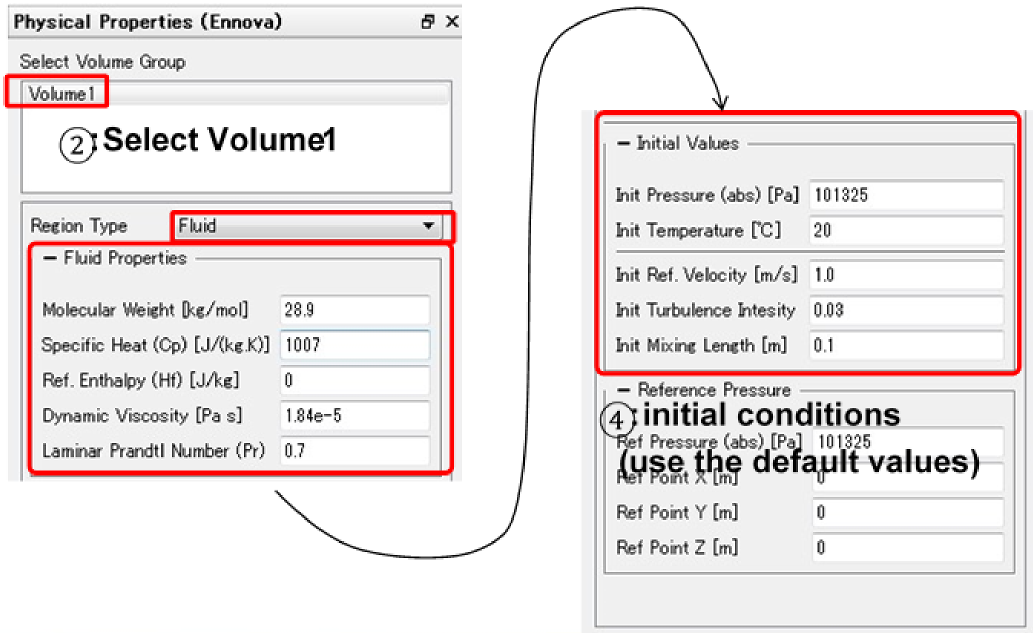
- Time :Stead State
- Conjugate Heat Transfer :Off
- Energy Equation :On
- Compressibility :Compressible
- Buoyancy :Off
- Radiation : On
- Solar Radiation : Off
- Turbulence Model :RANS
- RANS Model :Standard k-ε





1-2. Settings of the iconCFD solver

In the Physical Properties panel, the physical properties of the air will be set in this case.



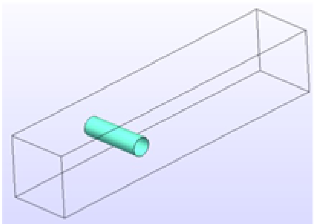
elect Fluid
the
perties of the
vill be set as
ult.

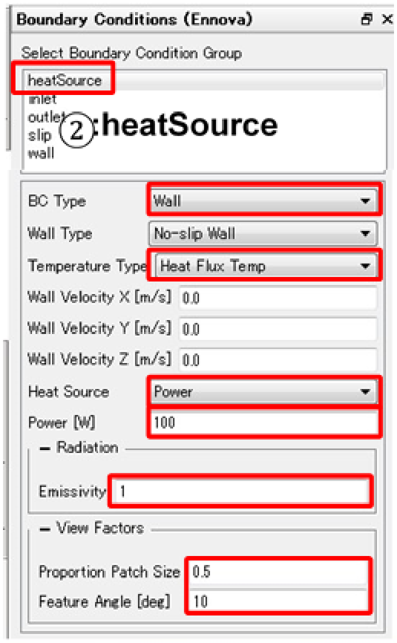
1-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel.

heatSource.

①





②:heatSource

③:BC type: wall

④:Heat Flux Temp

⑤:Flux: Power. The value is 100.

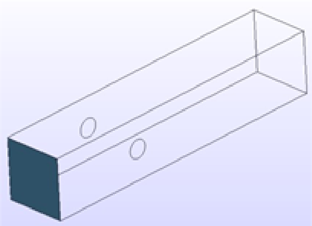
⑥:emissivity=1.0.
reflectance=1.0-emissivity

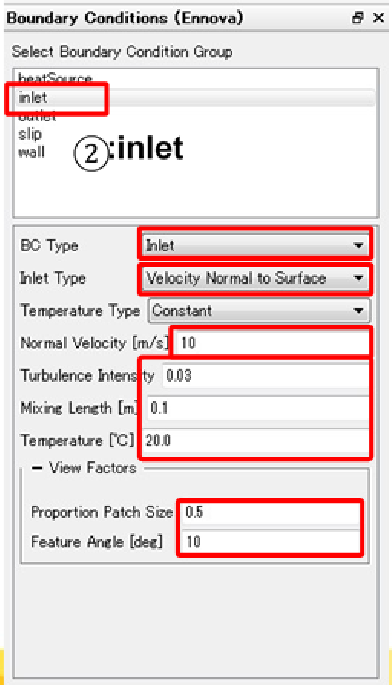
⑦:proportion patch size

1-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel (contd)

inlet





③:BC type: Inlet

④:Velocity Normal to Surface

⑤:velocity

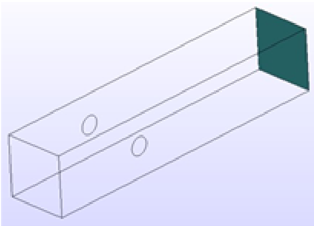
⑥:turbulence intensity and temperature as set as default.

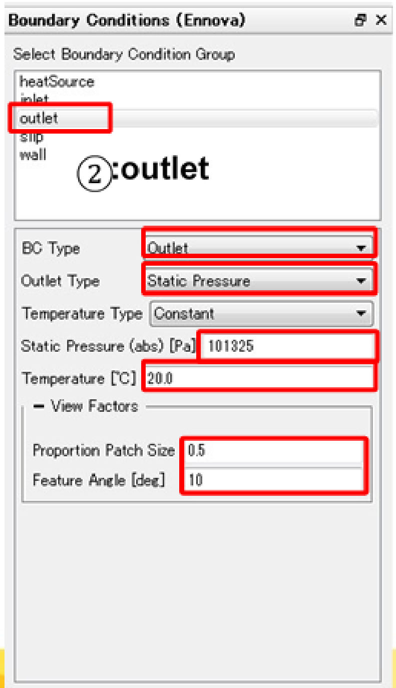
⑦:proportion patch size

1-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel (contd)

outlet





③:**BC type: outlet**

④:**Static Pressure**

⑤:**enter the pressure value**

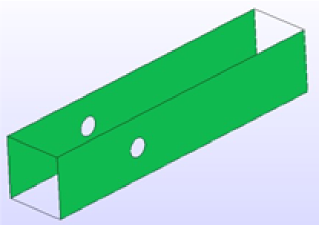
⑥:**enter the temperature**

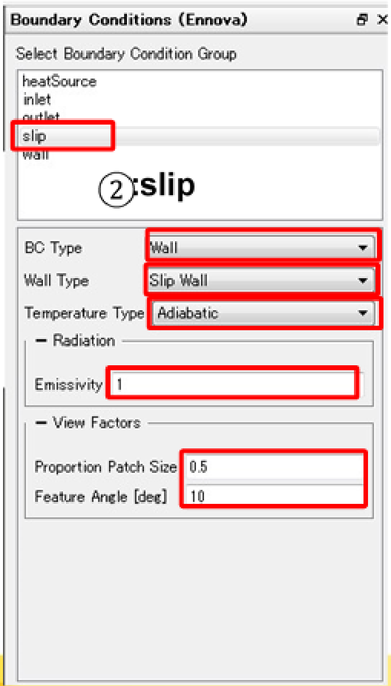
⑦:**proportion patch size**

1-2. Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel. (Contd)

slip





③:BC type: wall

④:Slip Wall

⑤:Adiabatic

⑥: Emissivity=1.0

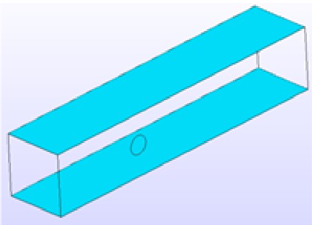
⑦:Proportion patch size

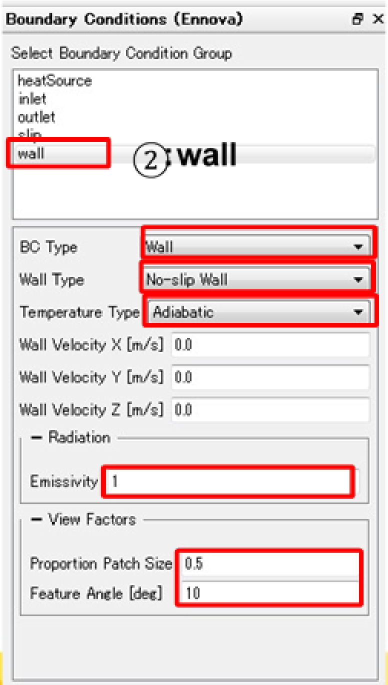
1-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel．(Contd)

wall

①





②:wall

③:BC type:Wall

④:No-Slip Wall

⑤:Adiabatic

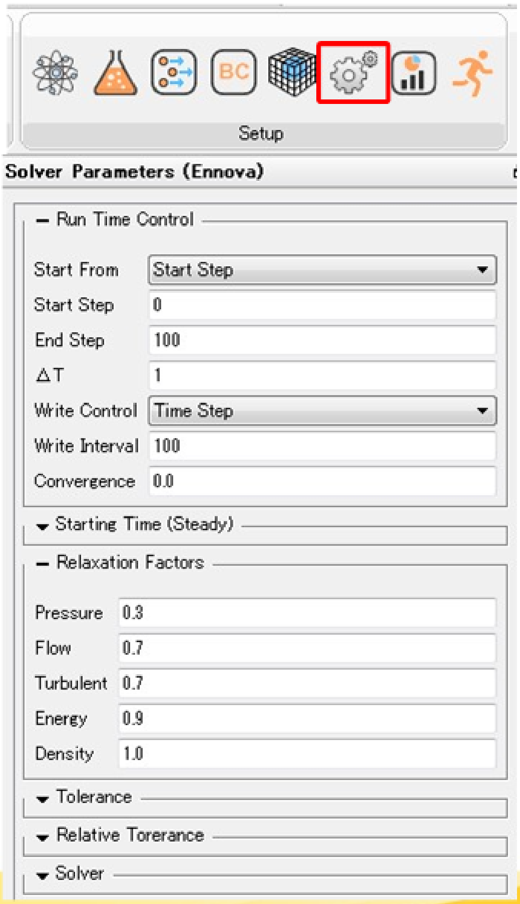
⑥:Emissivity=1.0

⑦:Proportion Patch Size

1-2 Settings of the iconCFD solver

Settings in the Solver Parameter panel.

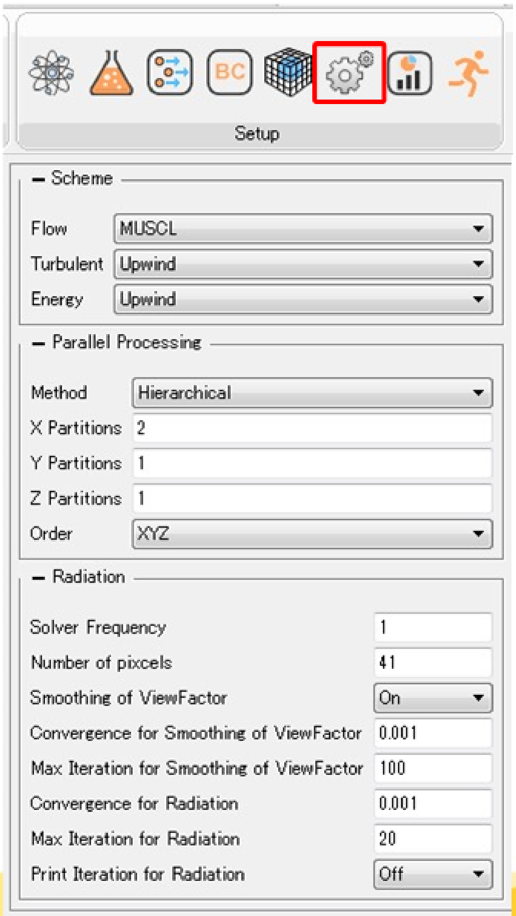
- Start Step/End Step
- ΔT
  - **Time interval**
- Write Control
- Write Interval
- Convergence
  - The criterion of convergence (in the case of 0,
- **simulation will persist until the specific iterations**)
- Relaxation Factor



1-2 Settings of the iconCFD solver

Settings in the Solver parameter panel. (Contd)

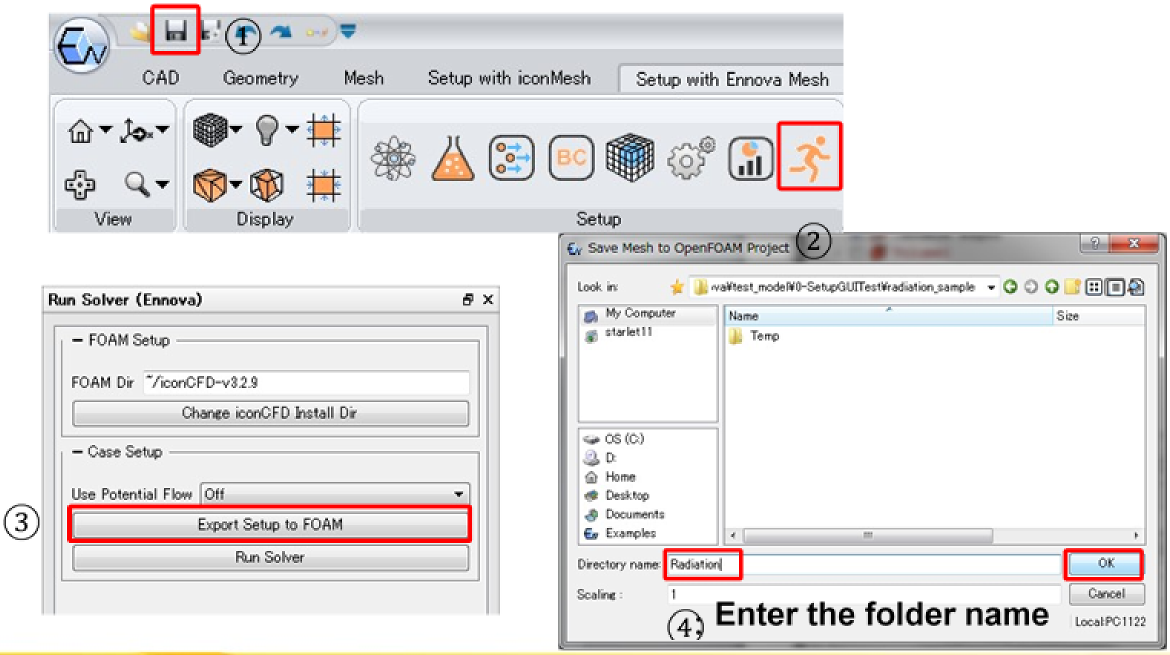
- Scheme
  - **Differencing scheme**
- Parallel Processing
  - **Parallel settings for iconCFD**
- Radiation
  - Solver Frequency
  - Number of pixcels
  - Smoothing of ViewFactor
  - Convergence for Smoothing of ViewFactor
  - Max Iteration for Smoothing of ViewFactor
  - Convergence for Radiation
  - Max Iteration for Radiation



1-2 Settings of the iconCFD solver

Saving the files for iconCFD solver

- Save the files set in ennovaCFD



1-3 Execution

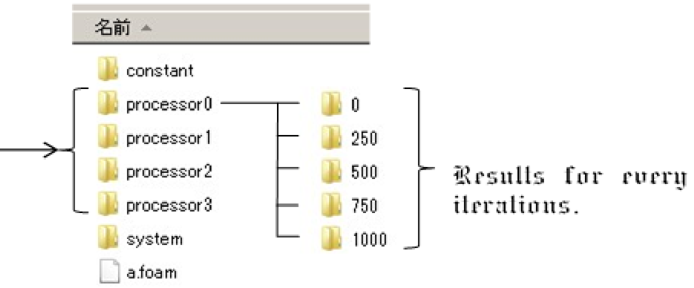
After the mesh and solver settings in the Windows terminal, copy the files to a system where iconCFD could run (such as simulation server).

Execute runHPC.sh to start simulations.

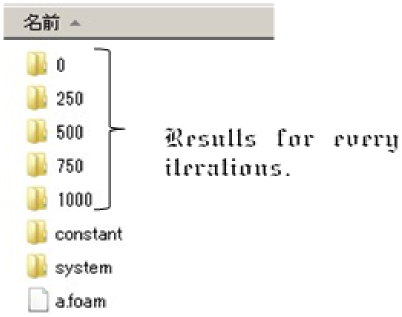
After the simulations, copy the results to the original terminal (windows).

The folders of the following kinds will be generated after the simulations are finished. The output frequency is as specified in the Solver Parameter panel.

- In the case of multiple processors



- In the case of single processor



•Results for each

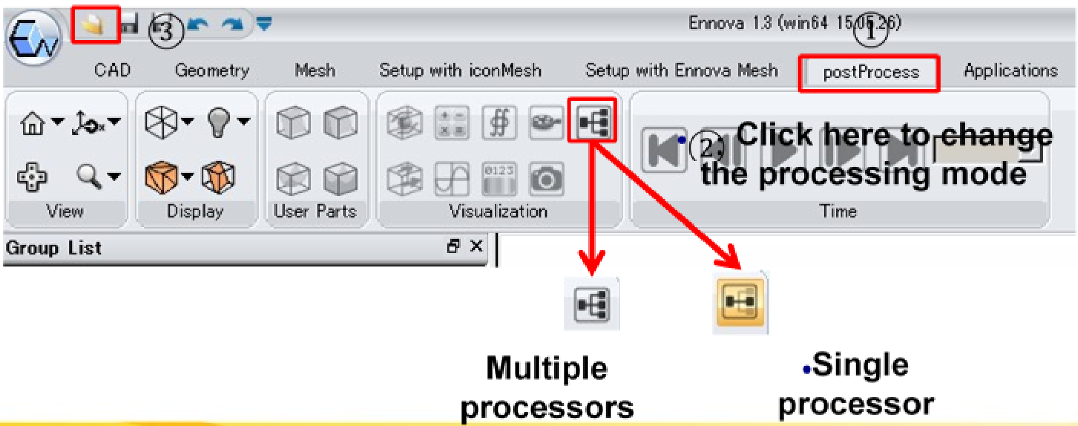
1-4 Postprocessing

Reading the simulation data from iconCFD (1/3)

- In the case of multiple processors
  - Put ***system*** , ***constant*** , ***processor**** , ****.foam*** files to a location accessible by Windows-PC.
- In the case of single processor
  - Put ***system*** , ***constant*** , ***time directories*** , ****.foam files*** to a location accessible by Windows-PC.

Selecting the postProcessing method

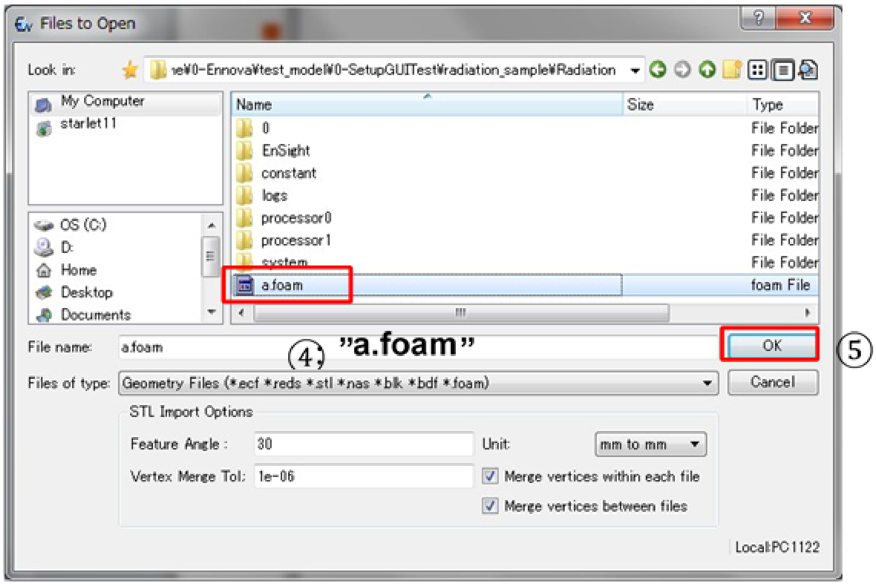
- Before reading the files, choose to process the results using multiple processors or single processor.



1-4 PostProcessing

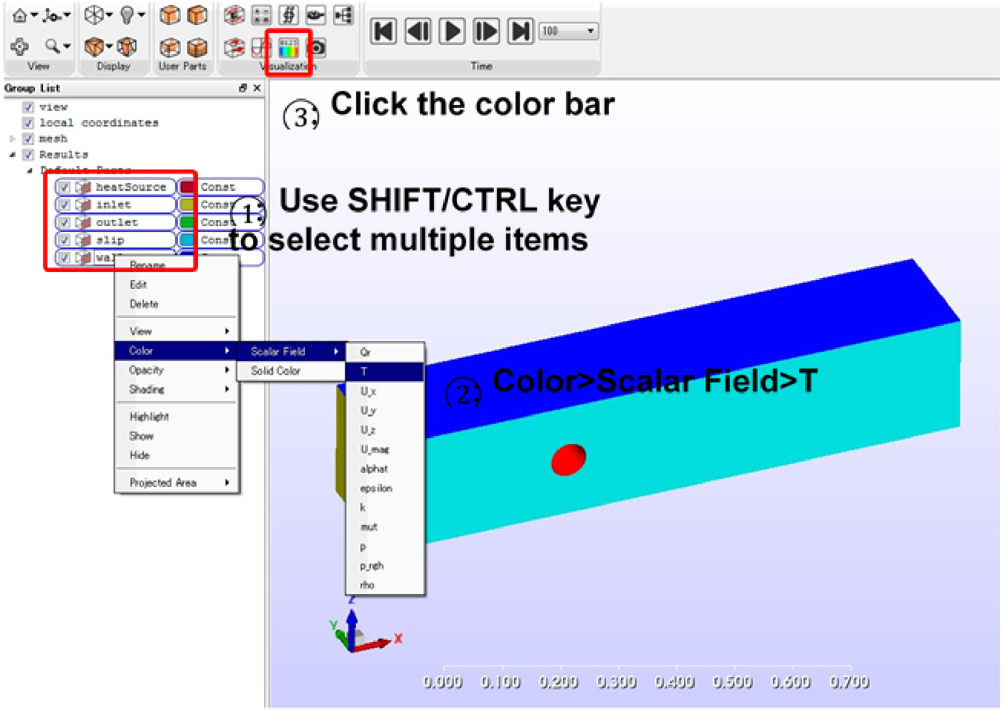
Read the output data of iconCFD (2/2)

- “a.foam” is automatically generated by runHPC.sh



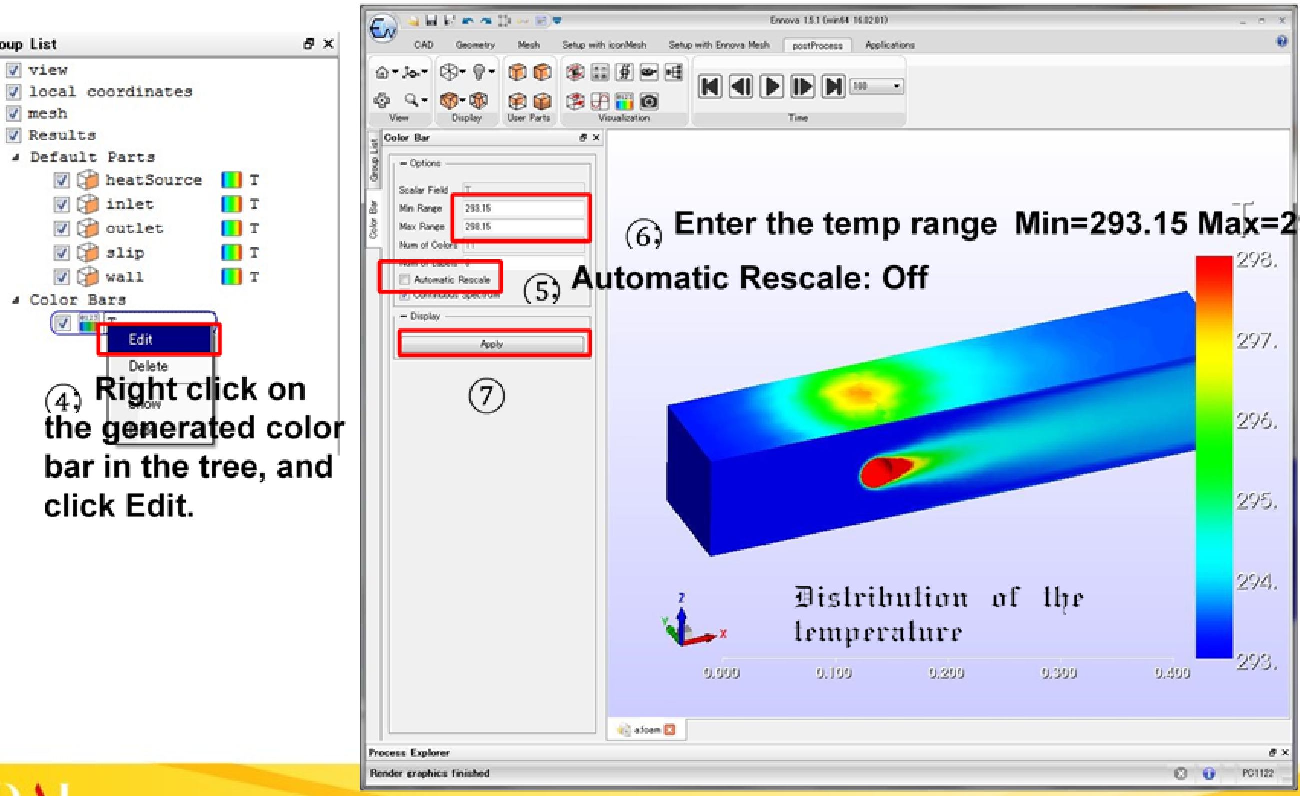
1-4 Postprocessing

Display the distribution of temperature K.



1-4 Postprocessing

Display the distribution of temperature K.

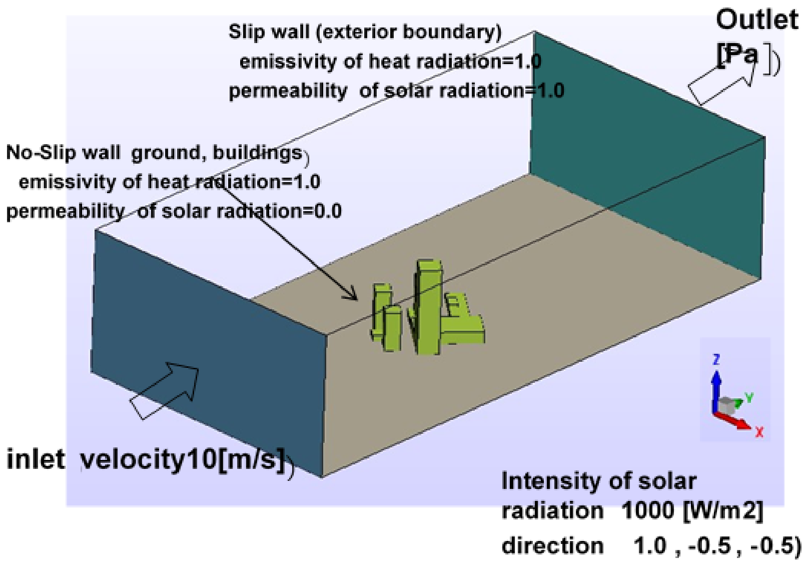


2. Settings for the solar radiation analysis

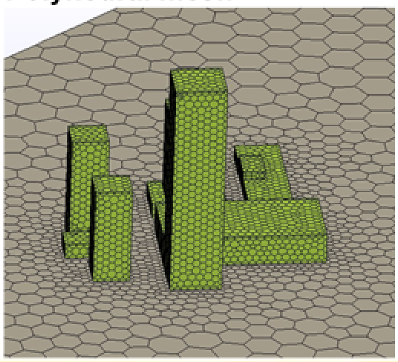
2-1. Introduction

We setup up the analysis of thermal radiation (S2S model) with ennovaCFD v1.5 and iconCFD3.2.11 in this manual.

The sol.ecf file in the sample data will be used. It is the meshed data (polyhedral mesh) generated by ennovaCFD.

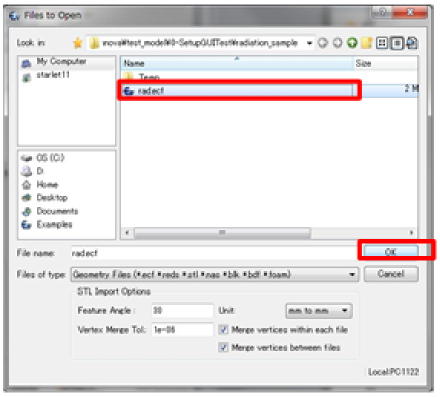


Polyhedral mesh

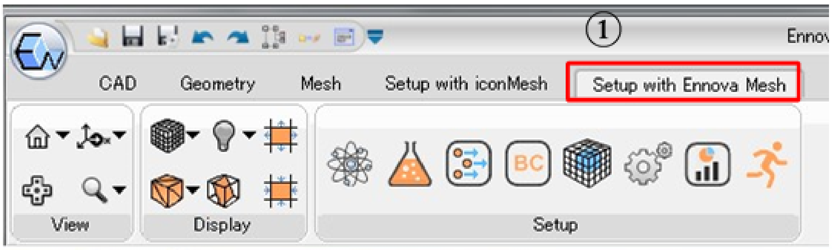


2-2 Settings of the iconCFD solver

Import the sol.ecf file.



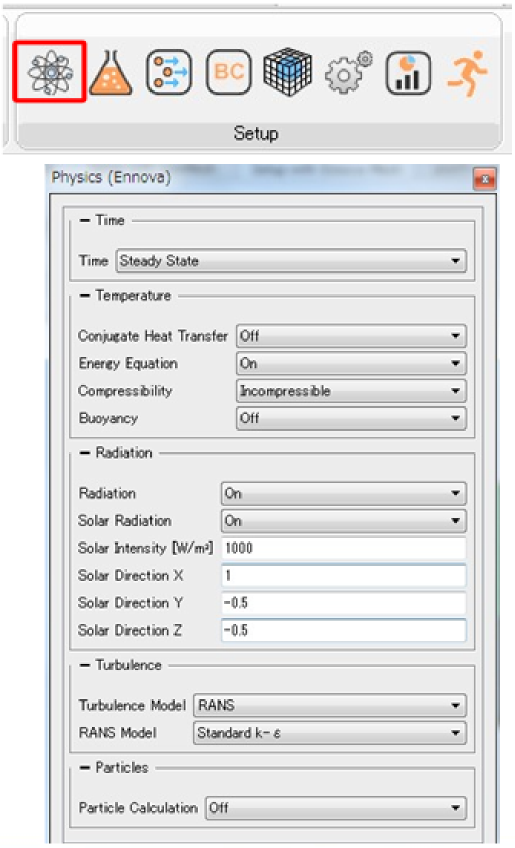
Since polyhedral mesh has already been generated by ennovaCFD, we proceed to Setup with Ennova Mesh.



2-2 Settings of the iconCFD solver

the Physics panel, steady/unsteady, mpressible/incompressible, temperature on/off, rbulence model, etc., will be specified. In the rrent case, steady state, compressible, diations will be set as the right figure.

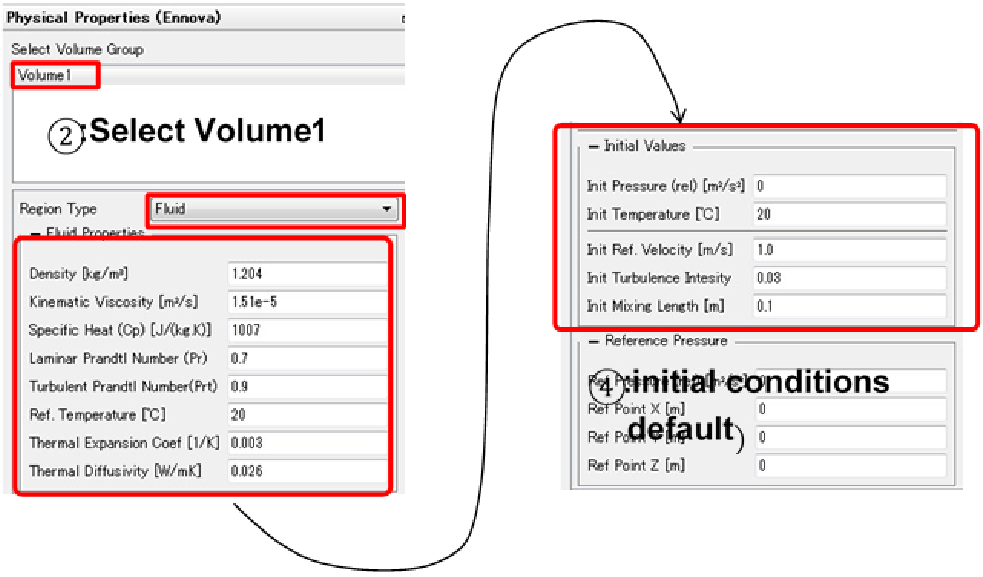
- Time :Stead State
- Conjugate Heat Transfer :Off
- Energy Equation :On
- Compressibility :Compressible
- Buoyancy :Off
- Radiation : On
- Solar Radiation : On
  - Solar Intensity
  - Solar Direction
- Turbulence Model :RANS
- RANS Model :Standard k-ε
- Particles :Off



2-2 Settings of the iconCFD solver

In the Physical Properties panel, the physical properties of the air will be set in this case.

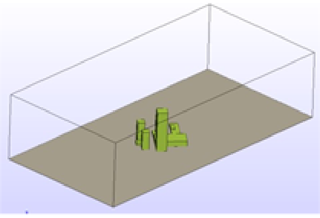
select Fluid
set the
sical
perties of air
ult)

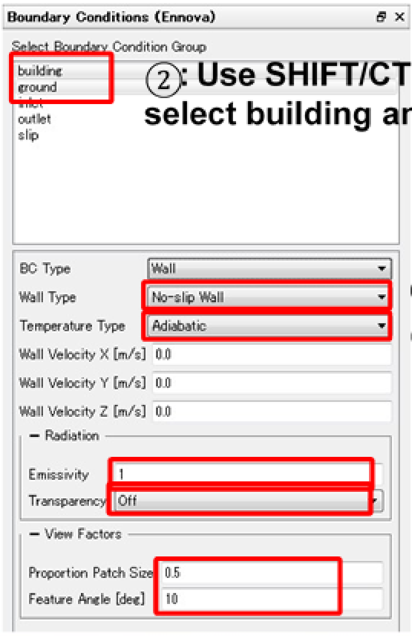


2-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel

building , ground: No-Slip, adiabatic





②: Use SHIFT/CTRL keys to select building and ground

③:No-Slip Wall

④:Adiabatic

⑤:emissivity of heat radiation=1.0
reflectance=1.0-emissivity

⑥:transparency: Off

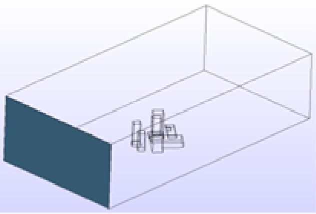
⑦:proportion patch size

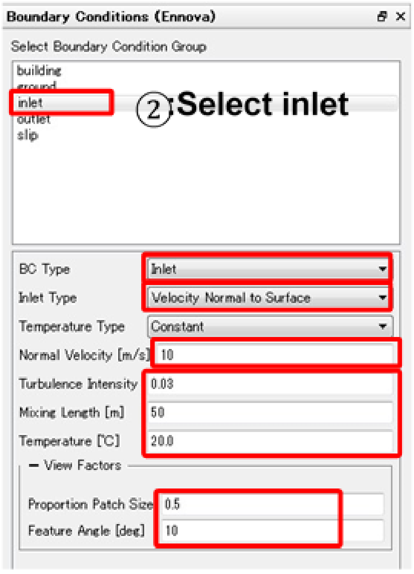
2-2 Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel.

inlet

①





②:Select inlet

③:BC type: Inlet

④:Velocity Normal to Surface

⑤:10[m/s]

⑥: turbulence intensity and temperature as set as default.

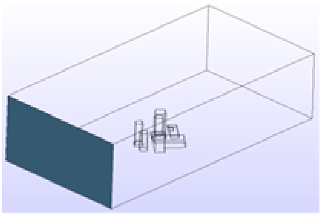
⑦:proportion patch size

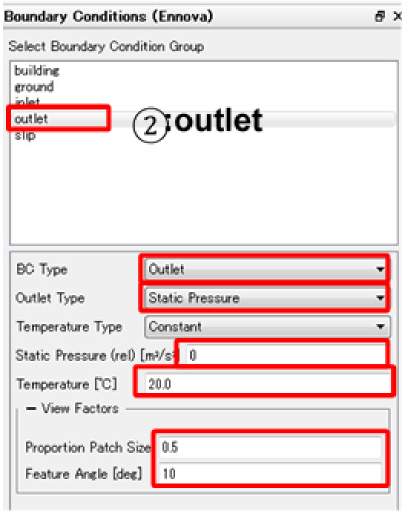
2-2. Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel.

①

outlet





②: outlet

③: BC type: Outlet

④: Static Pressure

⑤: 0[Pa]

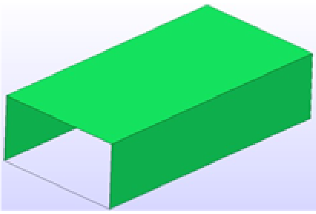
⑥: 20[℃]

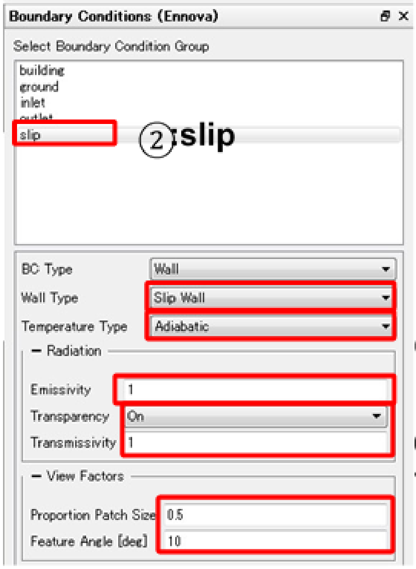
⑦: Proportion patch size

2-2. Settings of the iconCFD solver

Set the boundary conditions in the Boundary Conditions panel.

slip





③:Slip Wall

④:Adiabatic

⑤: Emissivity=1.0

⑥:Transparency: On

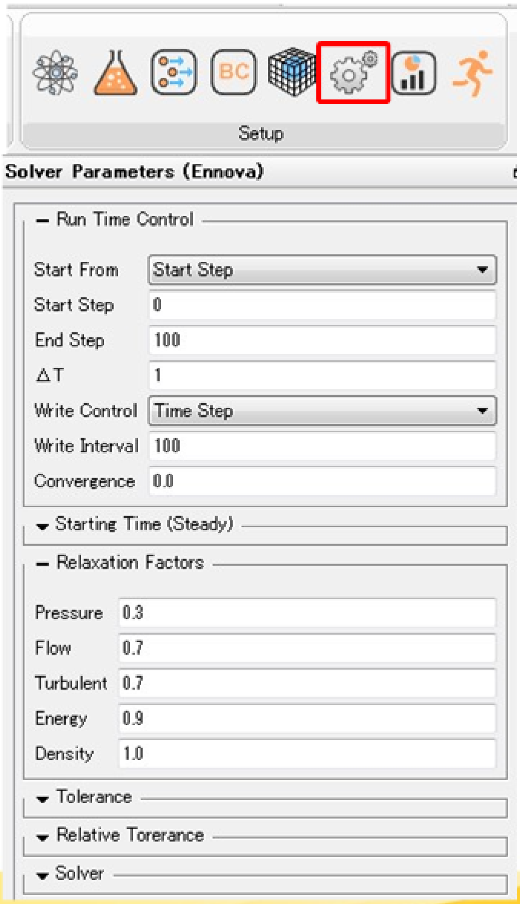
Transmissivity: 1.0

⑦:Proportion patch size

2-2 Settings of the iconCFD solver

Settings in the Solver Parameter panel.

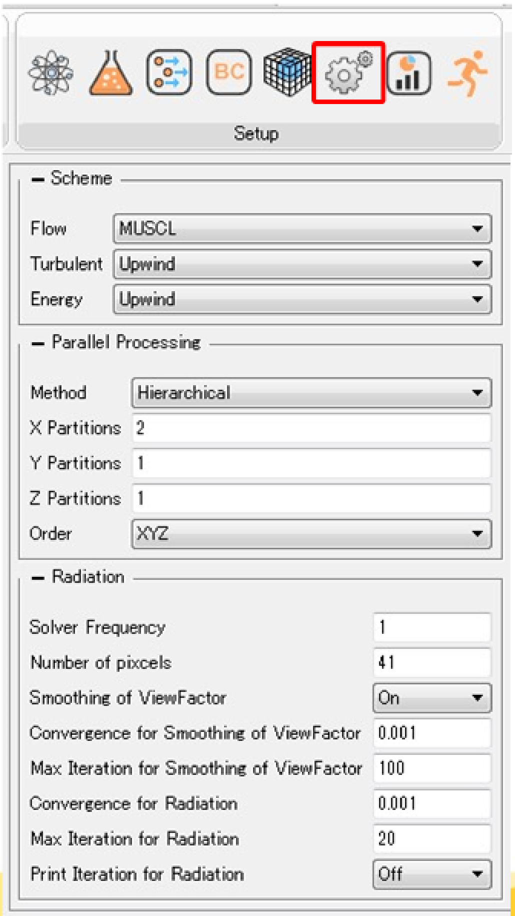
- Start Step/End Step
- ΔT
  - **Time interval**
- Write Control
- Write Interval
- Convergence
  - The criterion of convergence (in the case of 0, simulation will persist until the specific iterations)
- Relaxation Factor



2-2 Settings of the iconCFD solver

Settings in the Solver parameter panel. (Contd)

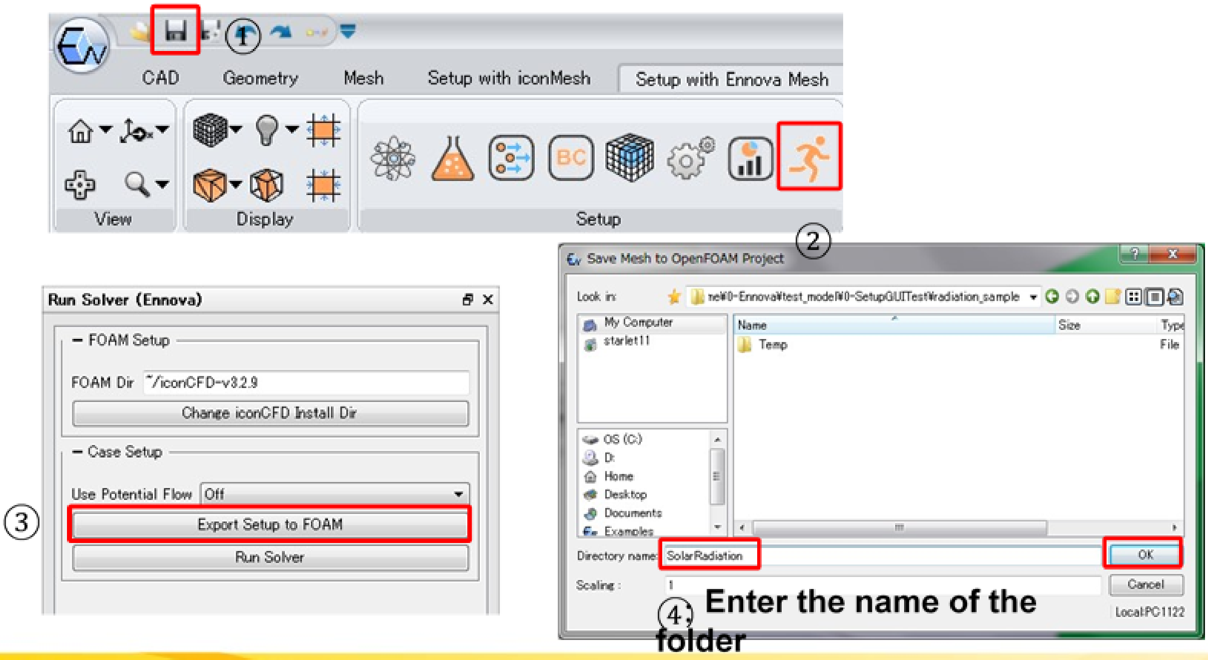
- Scheme
  - **Differencing scheme**
- Parallel Processing
  - **Parallel settings for iconCFD**
- Radiation
  - Solver Frequency
  - Number of pixcels
  - Smoothing of ViewFactor
  - Convergence for Smoothing of ViewFactor
  - Max Iteration for Smoothing of ViewFactor
  - Convergence for Radiation
  - Max Iteration for Radiation



2-2 Settings of the iconCFD solver

Saving the files for iconCFD solver

- Save the files set in ennovaCFD



2-3 Execution

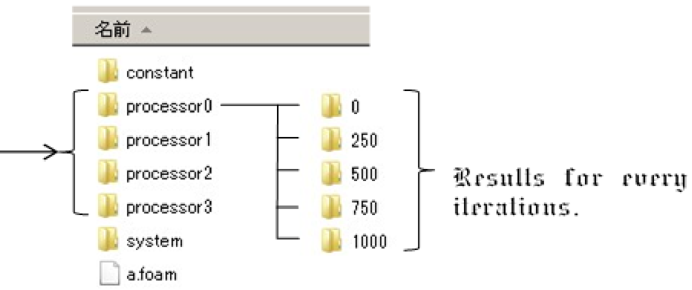
After the mesh and solver settings in the Windows terminal, copy the files to a system where iconCFD could run (such as simulation server).

Execute runHPC.sh to start simulations.

After the simulations, copy the results to the original terminal (windows).

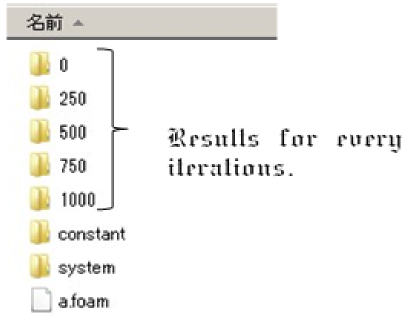
The folders of the following kinds will be generated after the simulations are finished. The output frequency is as specified in the Solver Parameter panel.

- In the case of multiple processors



•Results for each

- In the case of single processor



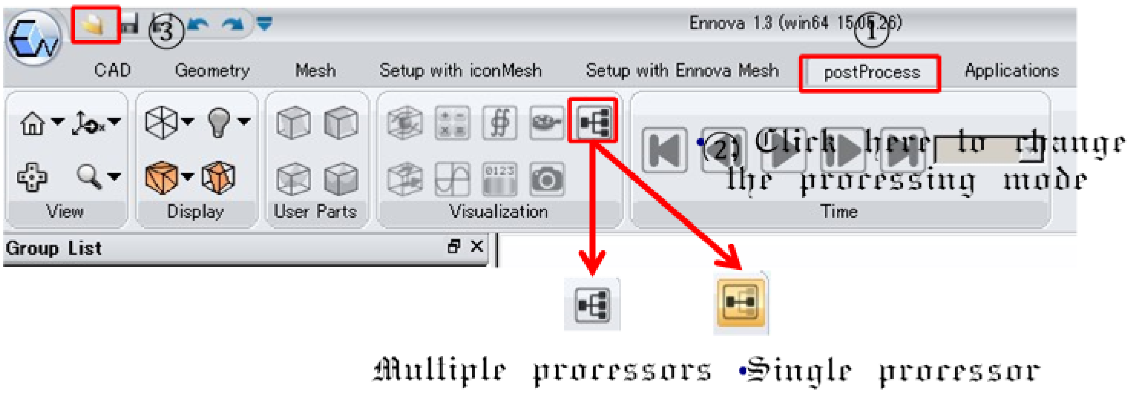
2-4 Postprocessing

Reading the simulation data from iconCFD (1/2)

- In the case of multiple processors
  - Put ***system constant processor* *.foam*** files to a location accessible by Windows-PC.
- In the case of single processor
  - Put ***system constant time directories *.foam files*** to a location accessible by Windows-PC.

Selecting the postProcessing method

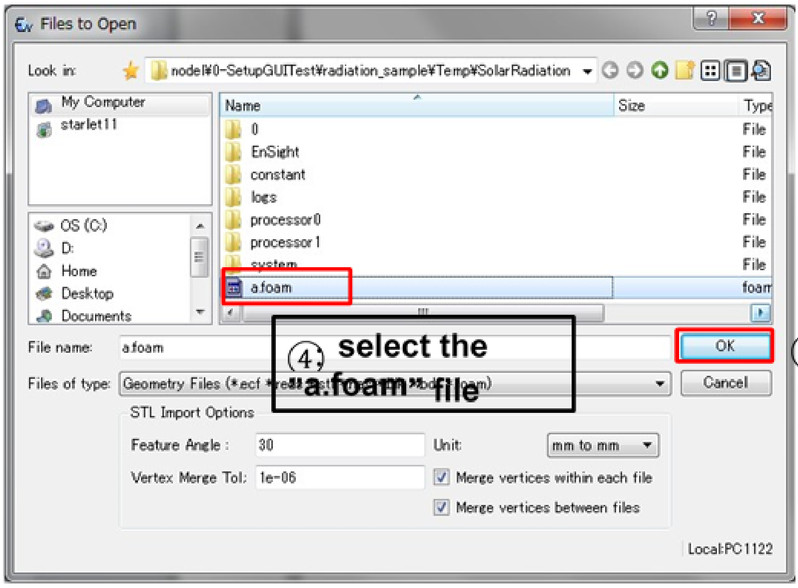
- Before reading the files, choose to process the results using multiple processors or single processor.



2-4 Postprocessing

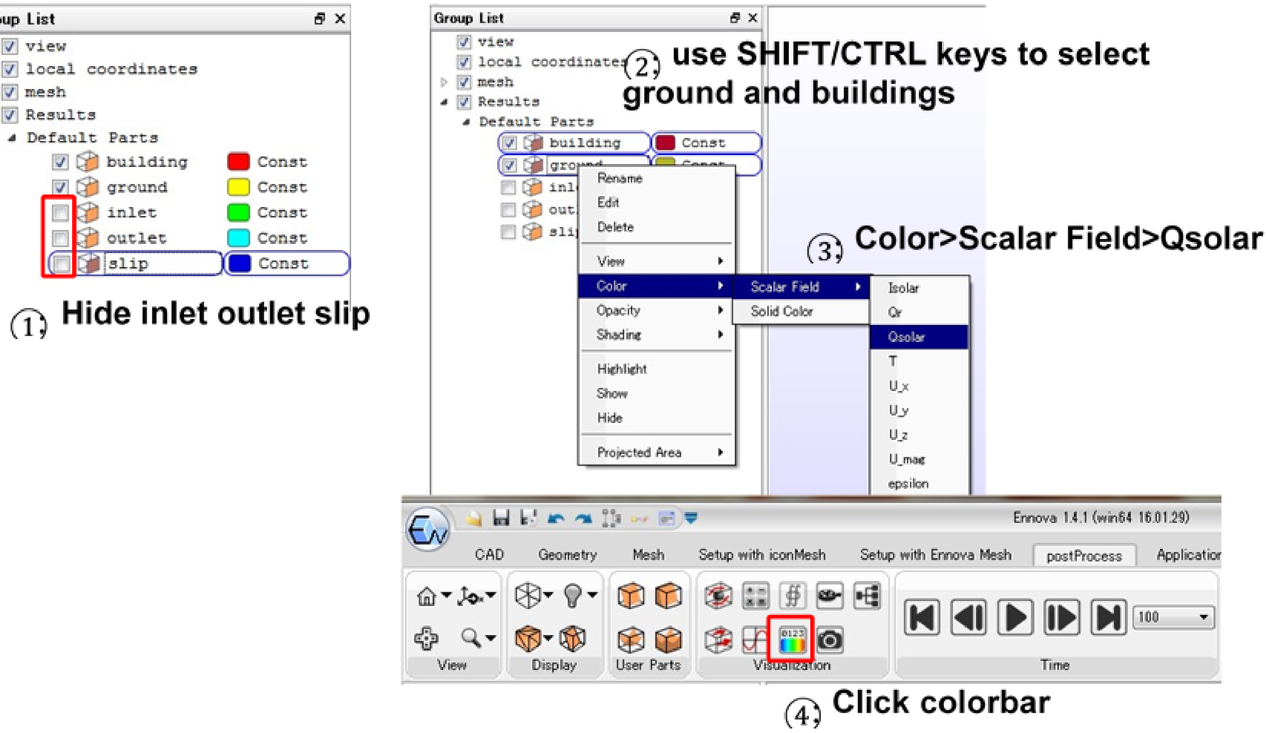
Read the data from iconCFD 2/2)

- "a.foam" is automatically generated by runHPC.sh



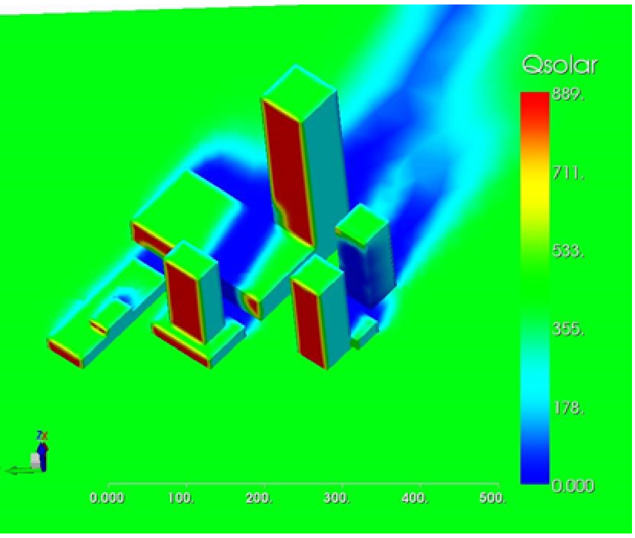
2-4 Postprocessing

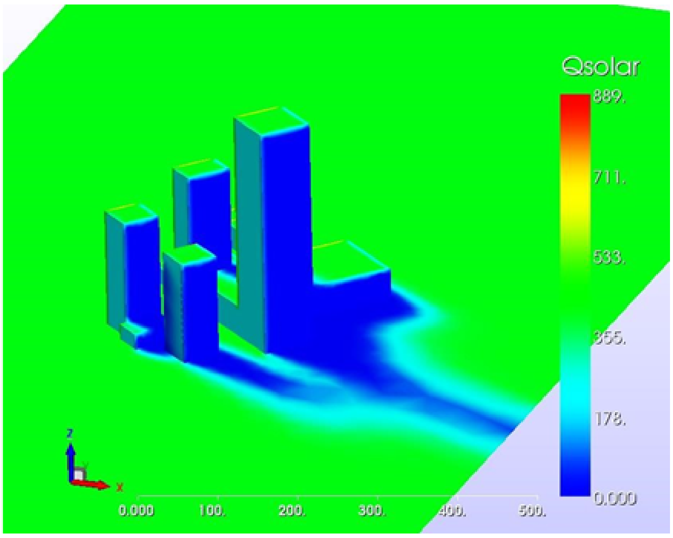
Distribution of solar radiation heat [W/m2]



2-4 Postprocessing

Distribution of solar radiation heat [W/m2]





Distribution of solar radiation heat

IDAJ
integrated digital advanced solution

Drivair Car Topology Mesh Tutorial

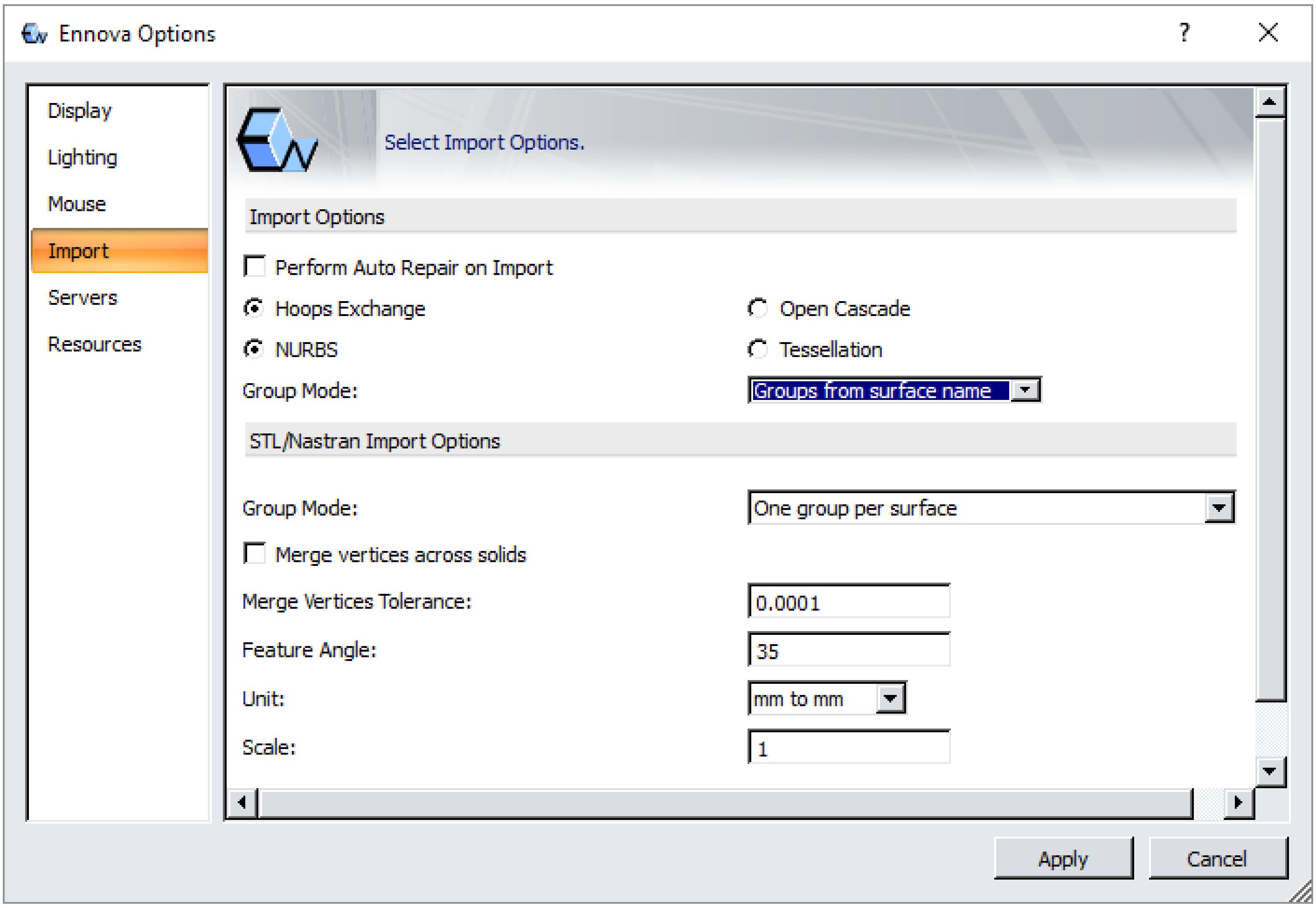
The DrivAer car is available after registering at this URL:
http://www.aer.mw.tum.de/en/research-groups/automotive/drivaer/

Once the download is complete, this tutorial uses the following parts:

01_Body.stp
02_Underbody_Detailed.stp
03_RearEnd_Fastback.stp
04_ExhaustSystem.stp
06_Wheels_Rear.step
05_Wheels_Front.step

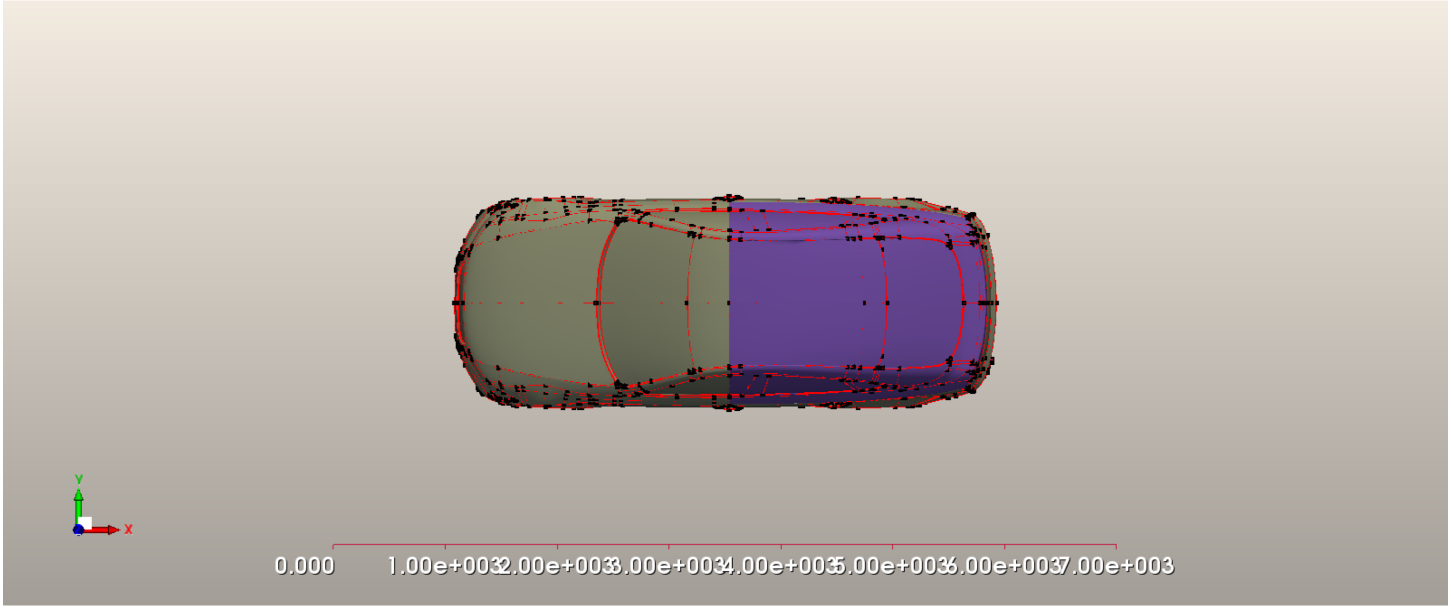
Ennova options

Use the Ennova Options to set the import mode to Groups from Surface Names (To get Options press the Blue Ennova logo in the top left hand corner)

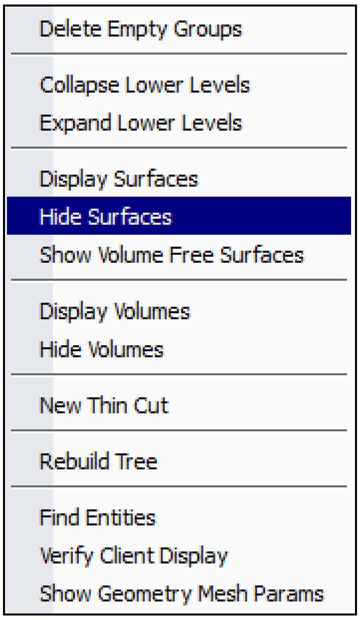


The CAD Model

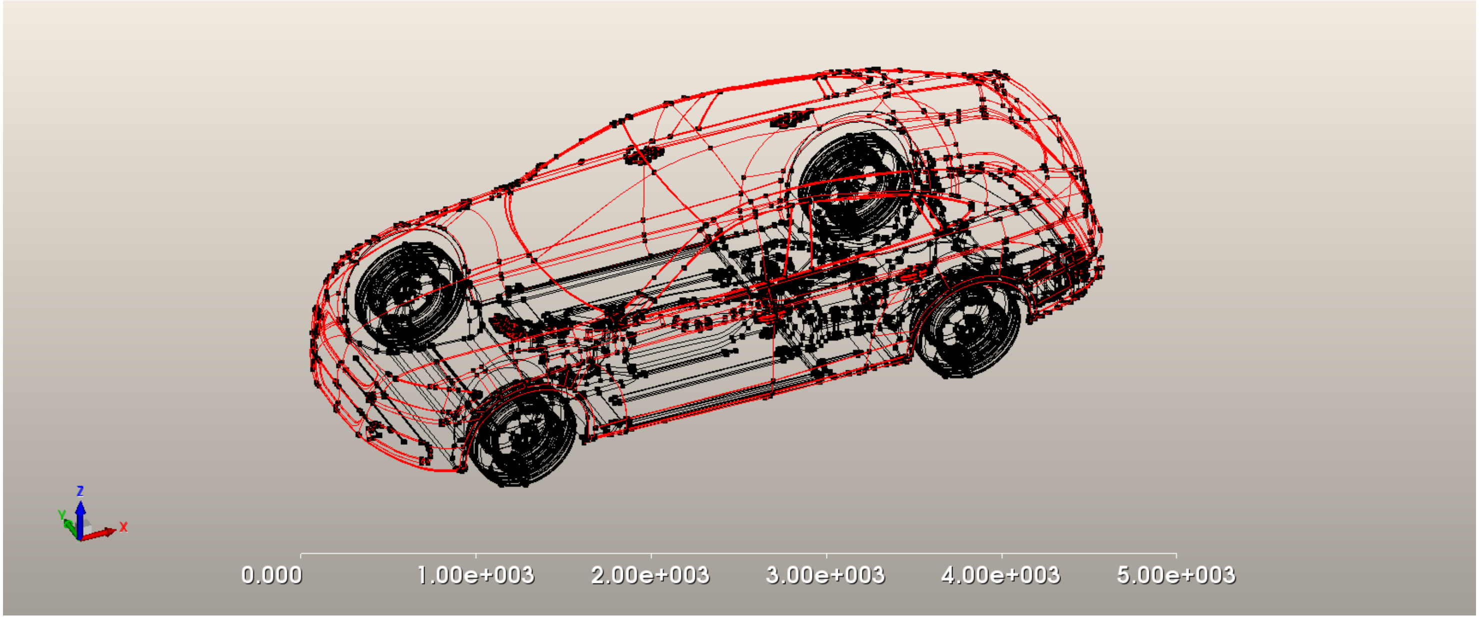
Loading the parts listed above will give the following model in Ennova:



Control the graphics by RMB on the geometry tree :

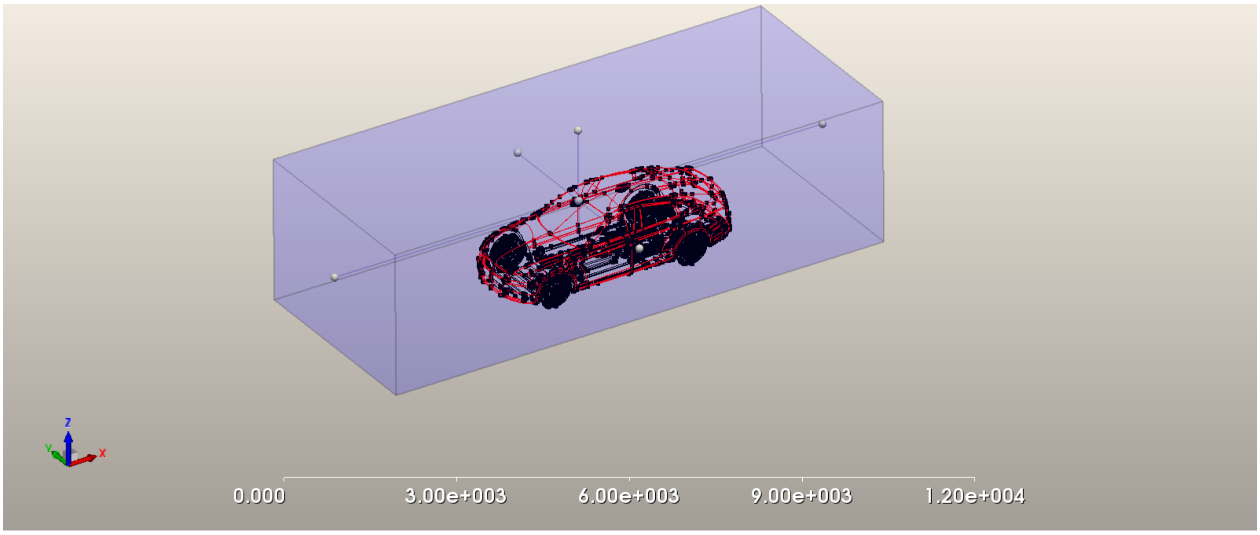


This will hide the surfaces and display the wire frame topology of the model

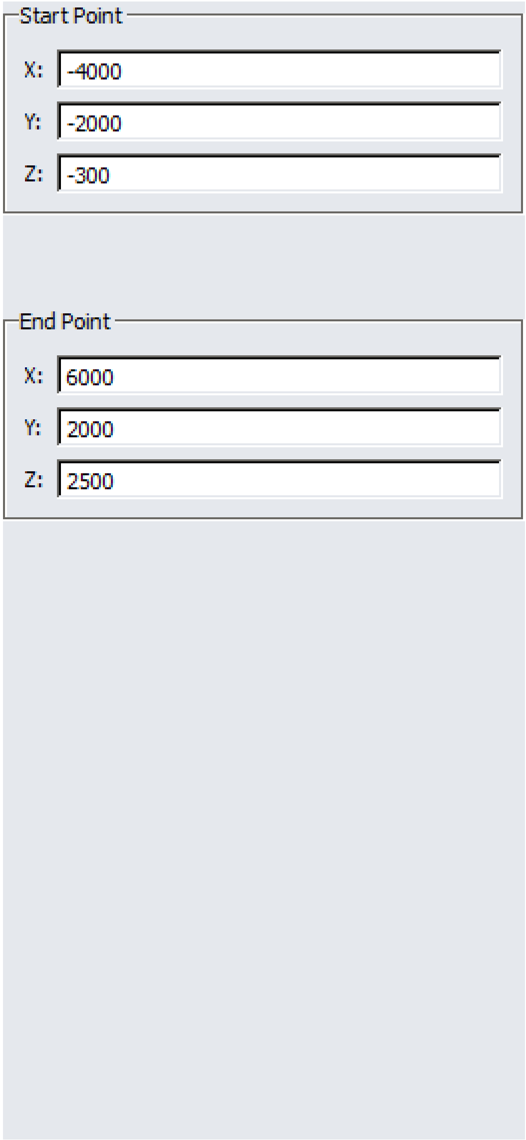


There are black and red edges. Red edges show the edge of a surface which is not stitched to its neighbour . Black edges show edges of surfaces which are stitched to another surface.
For an enclosed single volume with no non-manifold geometry (T intersections) the model needs to be repaired to be all black edges. But before we can repair the model we need a computational domain. In this case we can use a box to model a windtunnel. For the purposes of the tutorial we will make the windtunnel computational domain very small but for a real CFD analysis it shoud stretch at least 6- 10 car lengths upstream and 16 or more downstream.

Select the box  tocreate a box to create the windtunnel.

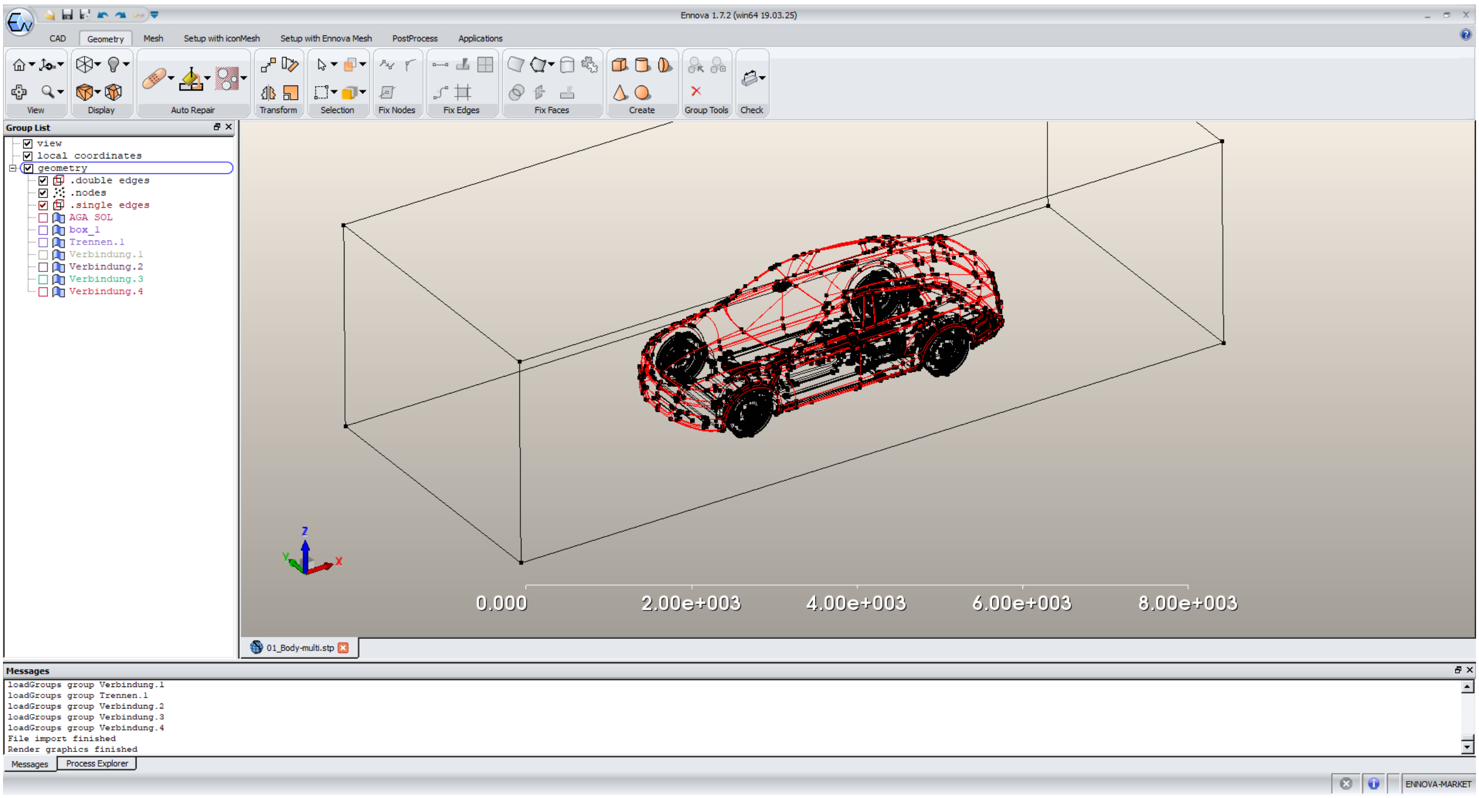


Adjust thesize either by typing in the numbers or dragging the box controls. For this tutorial use the following sizing:



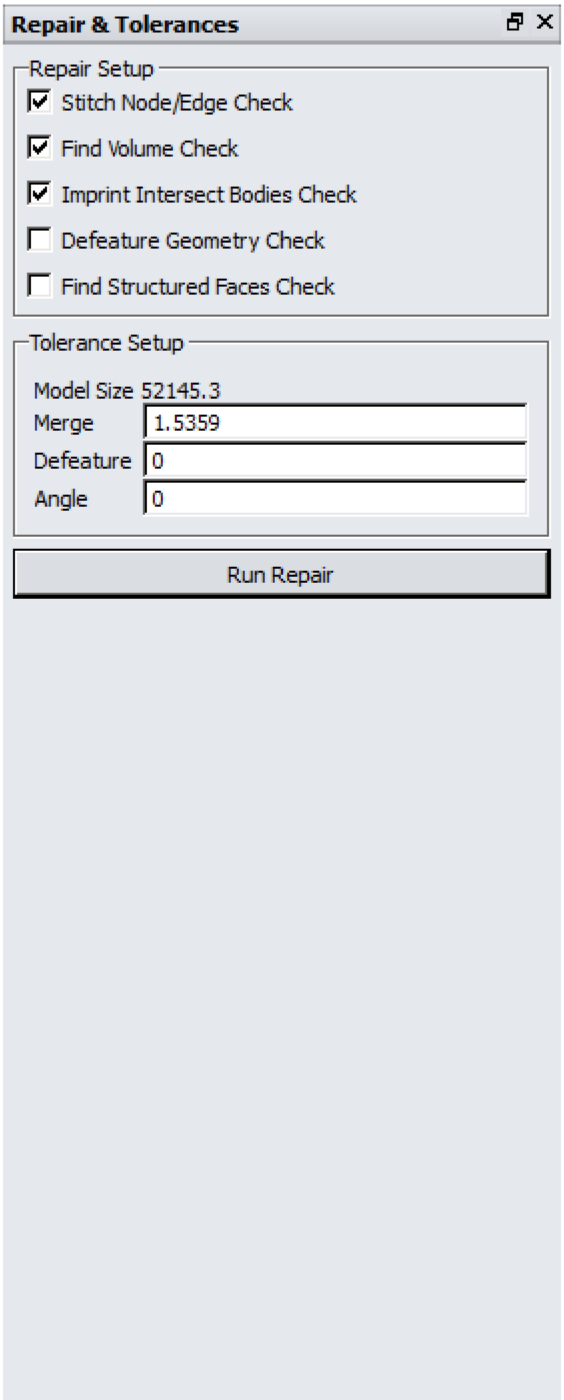
Press Add to complete

The Ennova display should now look like

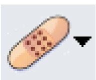


What we see are black double edges and red single edges. So the geometry is not all stitched together into volumes. We need to use the Ennova repair function to obtain our watertight computational volume.

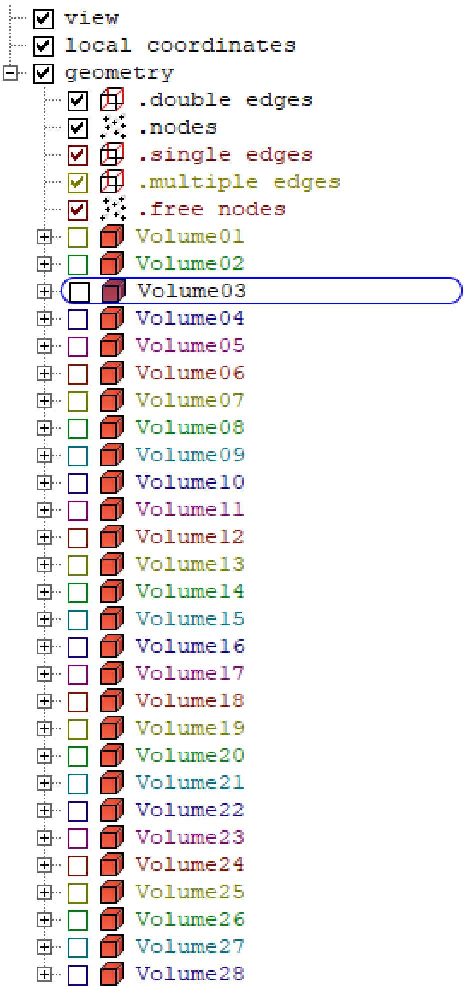
First we need to set the correct parameters for the repair function. Open up the repair menu by pressing the down arrow next to the bandaid in the menu ribbon



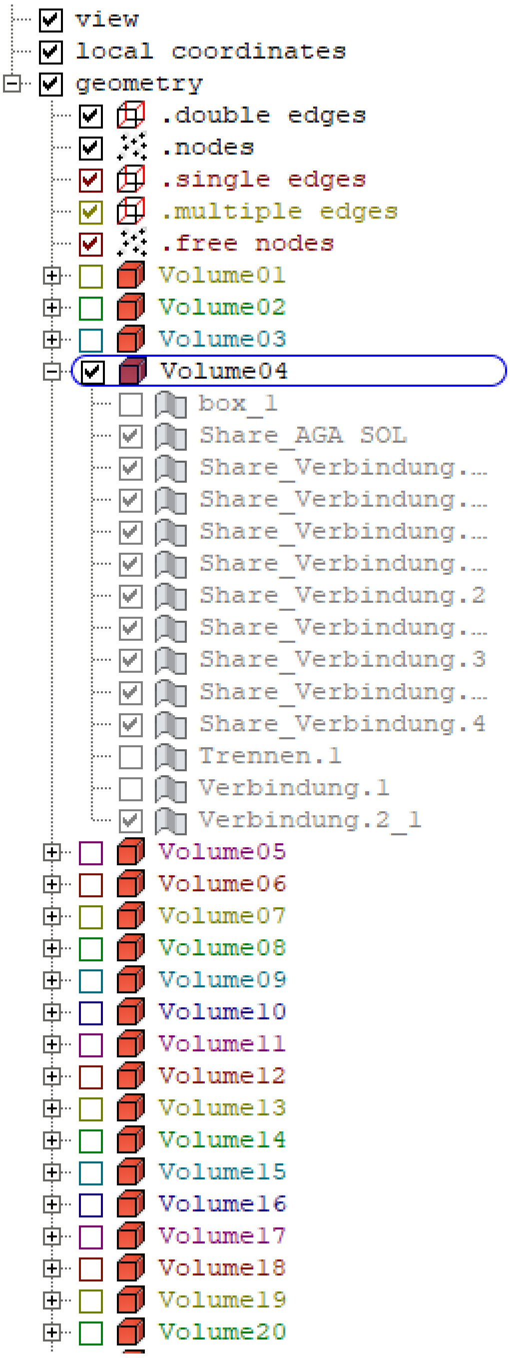
Accept the merge function tolerance of 1.54 mm and set the Defeature and Angle control to 0.000. Deselect the Defeature and Find structured Faces Check.

Press the  icon.

Ennova will begine repairing the model. You can observe the process in the message window

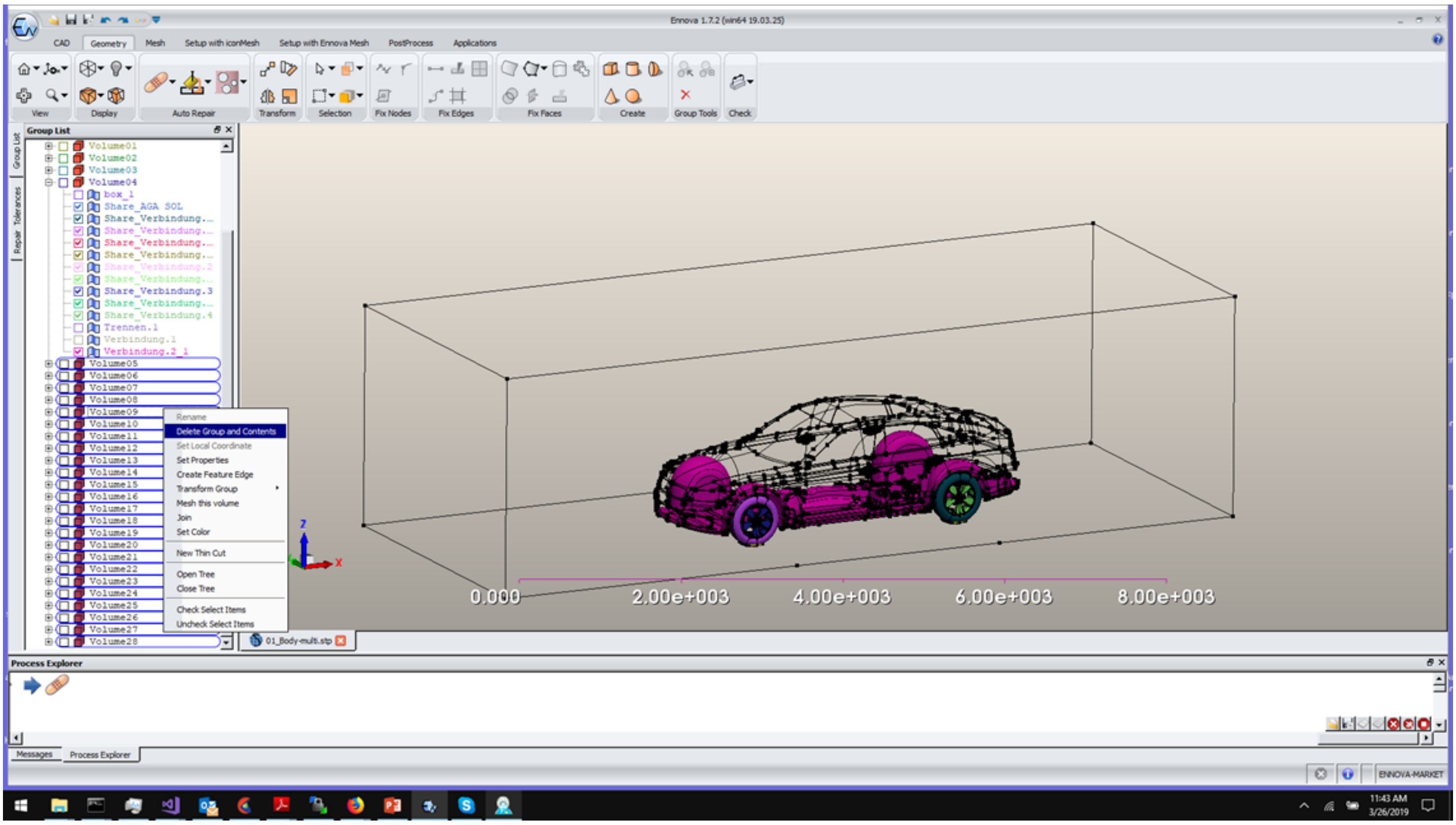


When complete the group list will report Ennova found 28 volumes.   By scrolling through the volumes visually we can identify volume 4 is the computational volume we require

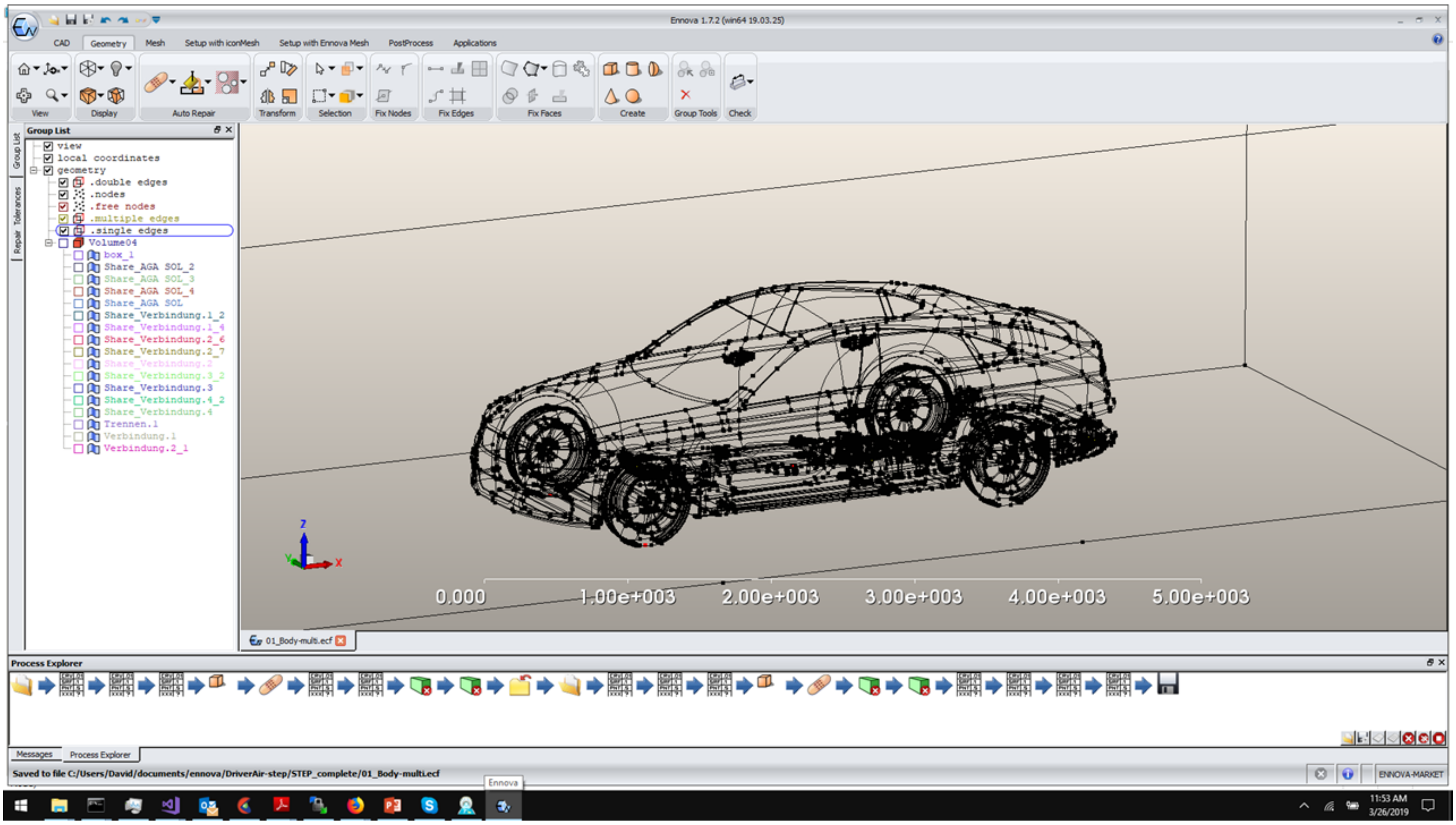


This includes the box and all the geometric parts

By Selecting the other volumes combined with a RMB we are able to delete them so as to be left with the correct computational volume



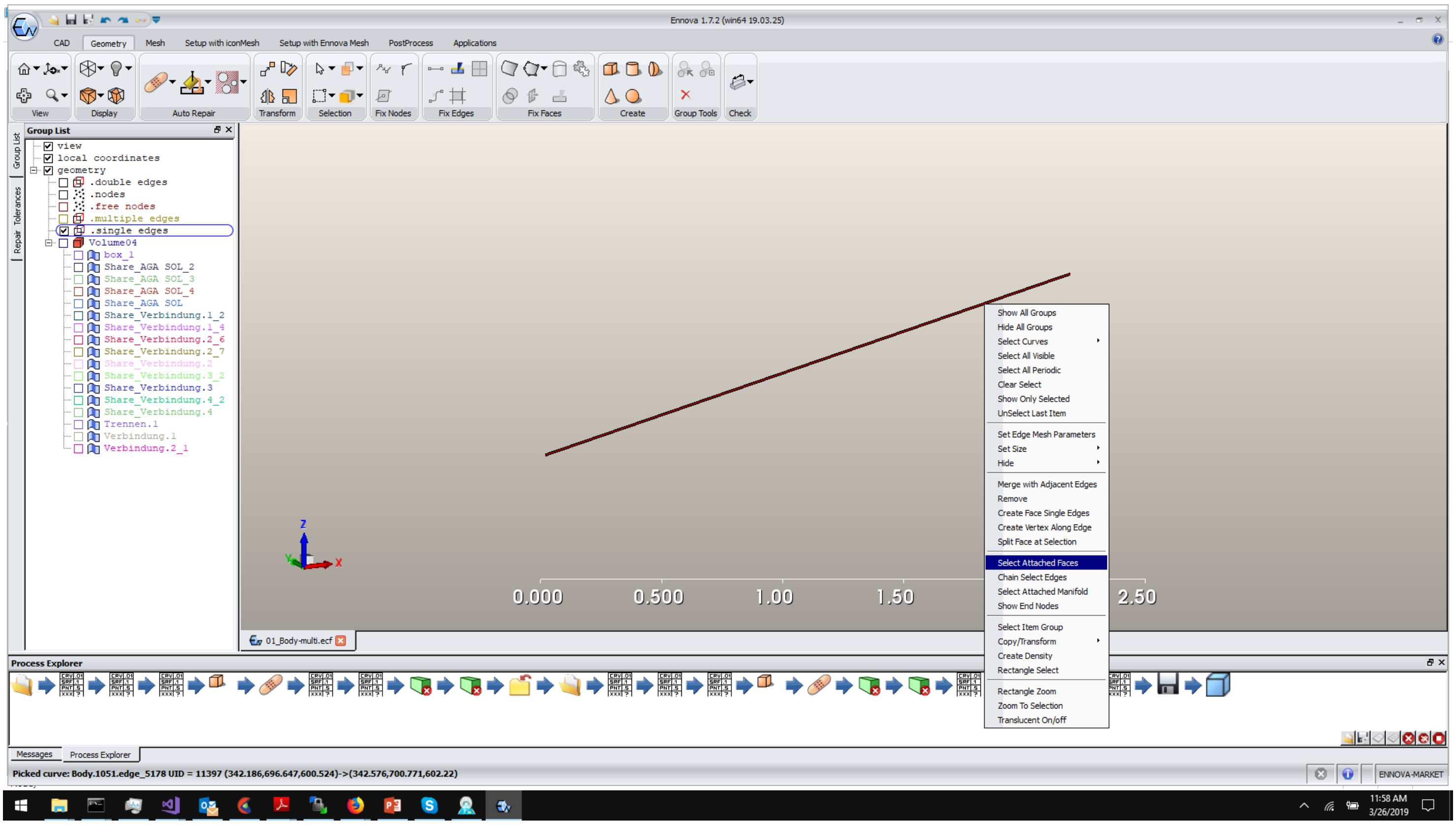
Now Our computational domain should look like this :



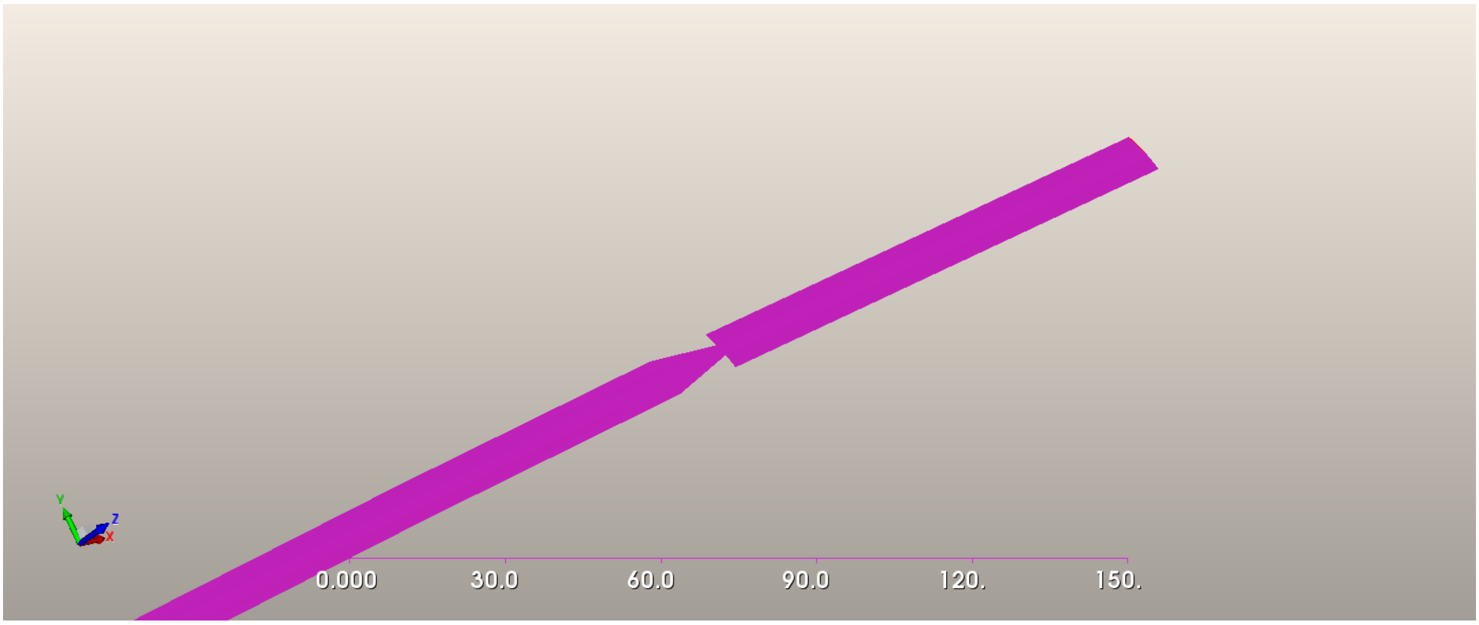
But if you look carefully we have some yellow edges and some red edges. If this was one manifold volume every surface edge would be connected to only one other surface edge and the whole model would be black. Yellow edges means that more than two (usually three) surfaces are connected at one edge. Red is as we already know indicates it is connected to a single surface.

We will now investigate and fix the yellow and red edges

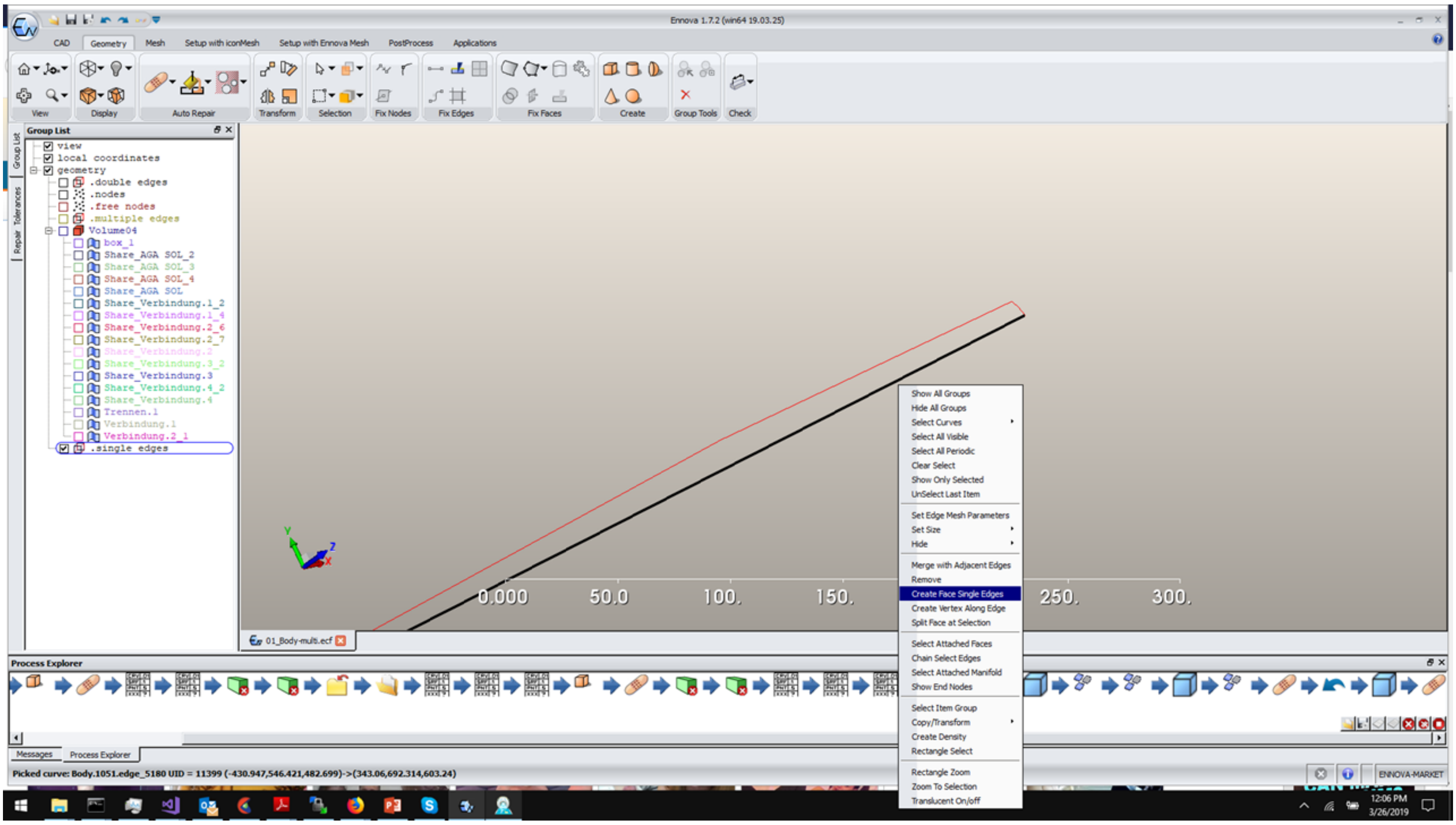
To do this turn off all the edges and press the home key. This will zoom to the red edge. By selecting the red edge RMB gives an option select attached faces.



After doing this we get a surface displayed.



Clearly this surface is incorrect (topologically). simply RMB and remove it. This will create a hole in your model of 4 red edges. Click on one of these edges and RMB. Select the option



This will create a new geometric face to fix the hole. There will be no more single edges in the model.

A similar approach can be done for the yellow edges.

Use the group list window to turn off all edges except the multiple Edges. Update the view using the Home key.

There will be several yellow (closed) loops in the model. These indicate where multiple faces are connected together. Wee wish to identify which surface are not needed for our volume and delete them to achieve a single manifold volume for this tutorial. There are many nuiances here do please watch the accompanying video for a detailed explanation of editing the geometry .

After the multiple edges have been removed your geometry should look like the following