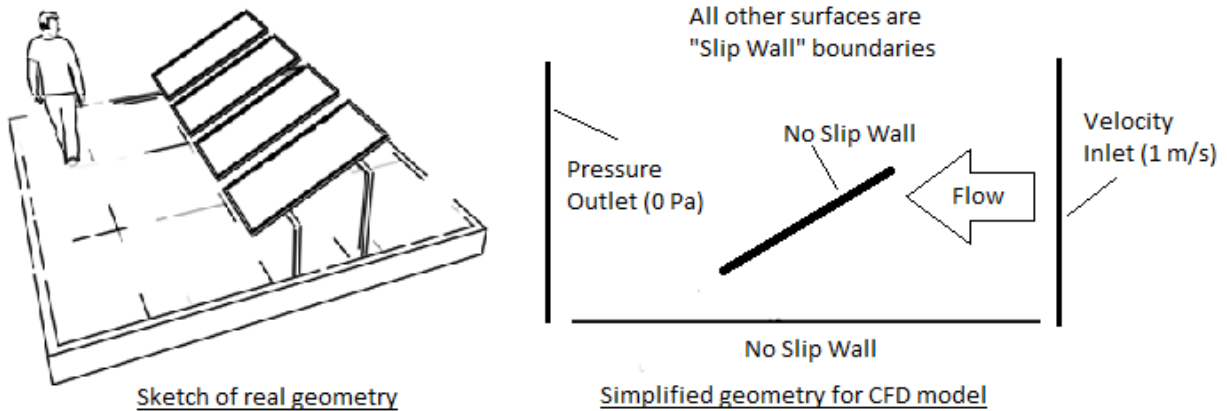


### Mesh Refinement for Solar Panel Airflow

**Note:** The STAR CCM+ File “scherba\_PVpanel\_MeshRefine.sim” is required for this tutorial.

In this tutorial you will be working with a fluid geometry that was imported from Solidworks. The geometry and all of the relevant flow characteristics are already set up for you. The purpose of this exercise is to demonstrate some mesh refinement techniques used to enhance the compatibility between the imported surface geometry and STAR CCM+. A specific problem relating to proper meshing of bodies in close proximity to each other is also addressed. Various meshes will be compared to each other, and two solutions will be generated. Results from each solution can be compared to show the impact of the mesh refinements.

The physical problem represented by this tutorial is a simplification of the flow regime surrounding a rooftop mounted photovoltaic solar panel (PV Panel).

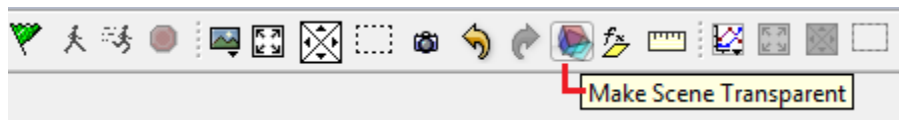


#### 1. Start STAR CCM+, open simulation file, view geometry and mesh scenes

File --> Load Simulation --> Browse --> 'scherba\_PVpanel\_MeshRefine.sim' --> OK  
Scenes --> Right Click 'Geometry Scene 1' --> Open  
Scenes --> Right Click 'Mesh Scene 1' --> Open

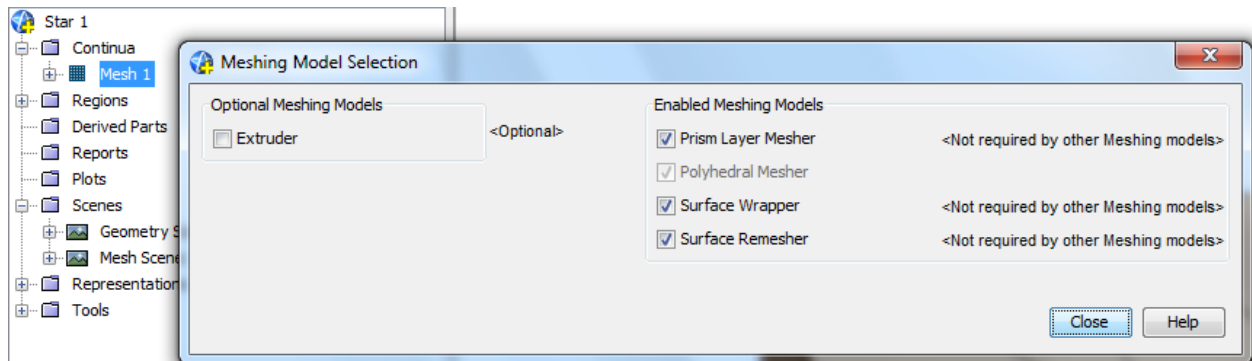
#### 2. Inspect the imported geometry

Select “*Make Scene Transparent*” from toolbar. You can now view the imported geometry, including the PV Panel internal to the fluid volume. Inspect both the geometry and mesh scenes. This initial mesh scene shows the triangular mesh that was automatically created with the imported geometry. As you can see, some mesh refinement is going to be necessary.



### 3. Assign Meshing Models

Continua --> Right click 'Mesh 1' -->'Select Meshing Models' --> Make selections as shown  
Continua--> Mesh 1--> Reference Values--> Base Size--> Confirm that the 'Base Size' is 1.0



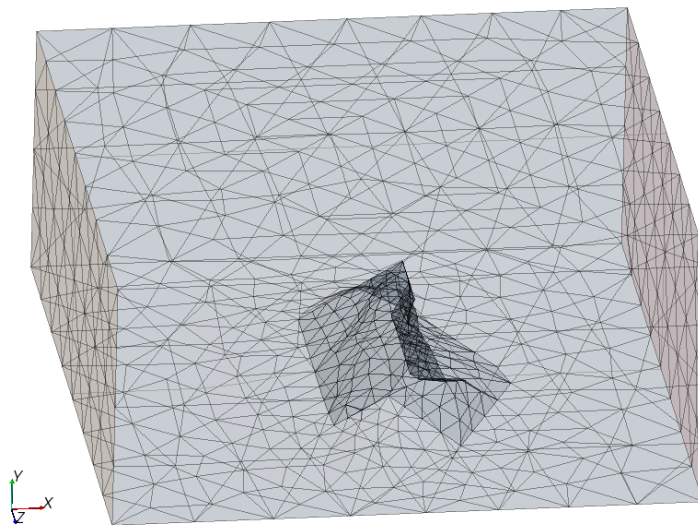
### 4. Generate surface mesh with default parameters

Mesh --> Generate Surface Mesh

### 5. Inspect surface mesh

Scenes -->Right Click Mesh Scene 1 --> Apply Representation --> Remeshed Surface

Select "*make scene transparent*" from the toolbar. Notice how the surface wrapper has merged the PV panel with the surrounding control volume. Obviously, further mesh refinement is still needed.



### 6. Refine surface size, Prism Layer Properties, and 'Tet/Poly Density'

Regions --> Region 1 -->Boundaries --> PV Panel --> Mesh Conditions -->Custom Surface Size -->  
Enable '*custom surface size*' with check box

Mesh Values -->Surface size --> Change '*Relative/Absolute*' to Absolute -->set '*Absolute Minimum Size*' to 0.05 and set '*Absolute Target Size*' to 0.1.

Continua --> Mesh 1 --> Reference Values --> Set the following:

*Prism Layer Thickness*: Relative Size = 5 % of base.

*Number of Prism Layers*: 3

*Prism Layer Stretching*: 1.5

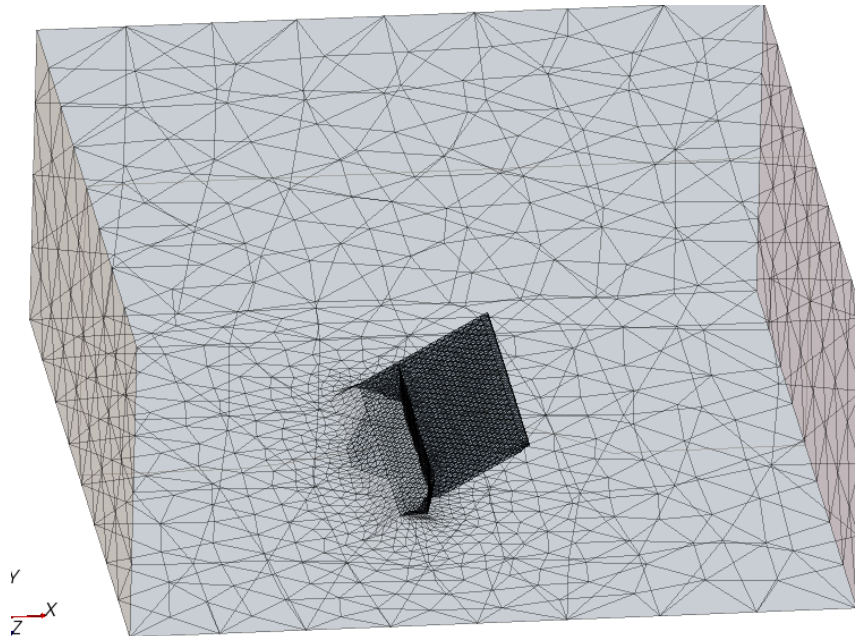
Continua --> Mesh 1 -->Reference Values-->Tet/Poly Density -->Set '*density*' to 1.0 and '*growth factor*' to 0.25. (This will decrease the growth of cells as you move away from the PV Panel)

#### 7. Generate Surface Mesh and Inspect, then Generate Volume Mesh

Mesh --> Generate Surface Mesh

Mesh --> Generate Volume Mesh

Note that most of the PV Panel is now well meshed, but the area closest to the control volume still needs some work.



#### 8. Run Solver to establish baseline results (these will later be compared to the better mesh)

Before starting the solution, it's helpful to set each scene representation to '*Volume Mesh*' so that the solution can be followed in the vector/scalar scenes.

Scenes--> Right click '*0.1 m above ground plane*'-->Apply Representation -->Volume mesh

Repeat for all scenes

Solution --> Run

Once residuals are low you can stop the solution.

## 9. Inspect Results

Right click 'Scenes' --> Open All Scenes

At this point you may want to save your simulation as a new filename, so that the results can be compared later.

## 10. Set Contact Prevention

Regions --> Region 1 --> Mesh Values --> Right Click 'Contact Prevention' --> New --> 'One Group Contact Prevention Set'

Select the newly created 'Contact Prevention 1' and set the following:

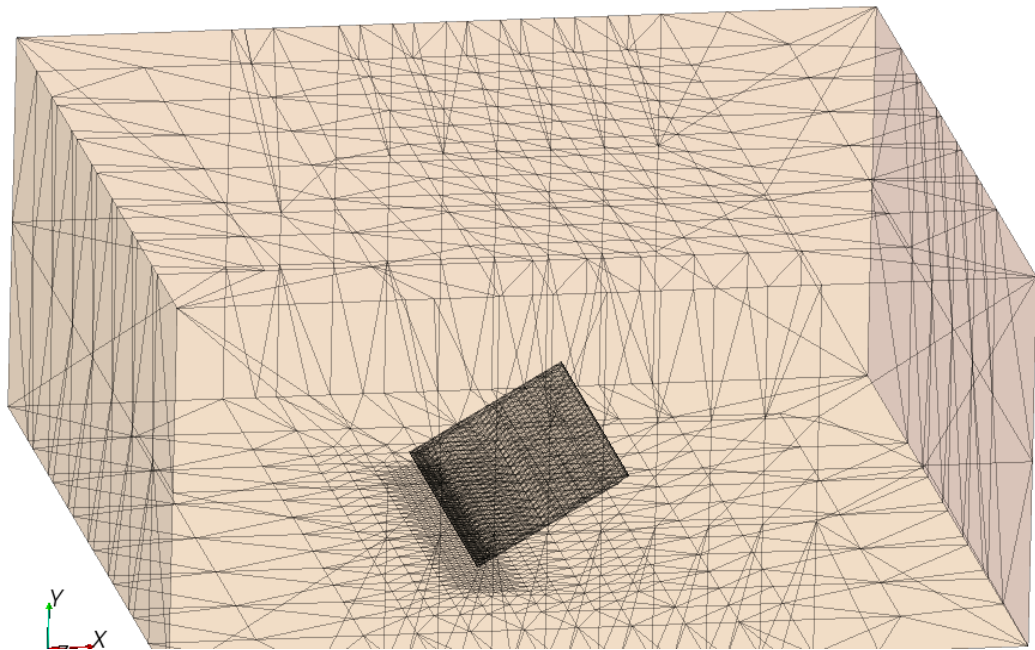
Boundaries: *Ground Surface , PV Panel*

Search Floor: 0.001 m (this parameter influences the number of cells that can be generated between the PV Panel and Control Volume)

## 11. Generate surface mesh with refined parameters and inspect

Mesh --> Generate Surface Mesh

This mesh is now refined enough to recognize the PV panel and control volume as separate regions.



## 12. Generate volume mesh and run Solver

Mesh --> Generate Volume Mesh

Using transparency or a section view you should be able to inspect the mesh and see that it has been refined in the region surrounding the PV panel. Further adjustments can be made to the PV Panel surface mesh size, prism layer properties, Control Volume surface mesh sizes, etc.

Scenes--> Right click '0.1 m above ground plane'-->Apply Representation -->Volume mesh

Repeat for all scenes

Solution --> Run

## 13. Compare Results

The figure below shows a comparison of the "0.1 m above ground plane" scene. This shows the velocity vectors on the plane mid-way between the lower edge of the PV panel and the ground. It can be seen that the flow field on this plane is drastically impacted by the changes that were made in the meshing. In the coarse mesh, flow is diverted around the PV panel, while in the fine mesh, flow is allowed to pass between the panel and the ground.

However, if you compare the two 3D streamline plots (not pictured here), you will notice that the overall flow regime is impacted by the mesh change, but not in an extreme manner. The mid-plane velocity vector plot shows that, with the refined mesh, there is an additional recirculation zone on the top face of the PV panel. This is an area that should be studied further, and could become more pronounced at higher inlet velocities. If your interest is on the local flow regime immediately surrounding the PV panel, the refined mesh should be used for any further analysis.

