Exploring inhaled air-particle-vapor mixtures in the Human Respiratory System with Ansys Fluent® simulations

Part 3: Pre-Processing Geometry Import and Mesh Generation using Ansys Fluent Meshing

Yu Feng, Ph.D.

Developed by CBBL at Oklahoma State University

Edited by the Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This tutorial uses Ansys Fluent®, the fluid simulation software.

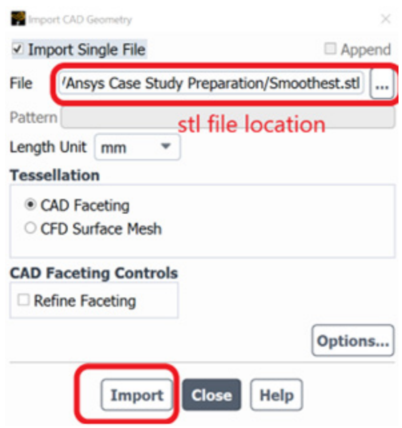
Learning Goals

In this tutorial segment, we will cover:

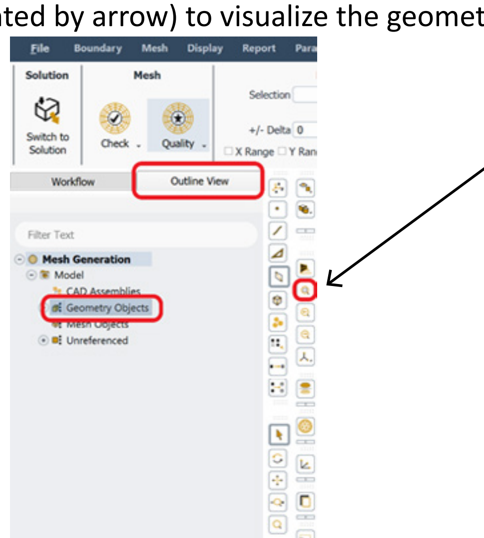
- Geometry import
- Mesh Size Determination
- Surface Mesh Generation
- Mesh Quality Check
- Material Point Creation and Volumetric Region Creation
- Volume Mesh Creation
- Volume Mesh Quality Check and Improvement
- Export Mesh

Import the Airway Geometry

Open the Ansys Fluent software[1] in Meshing Mode
Import stl file: File→Import→CAD→Load the stl file→Click "Import"

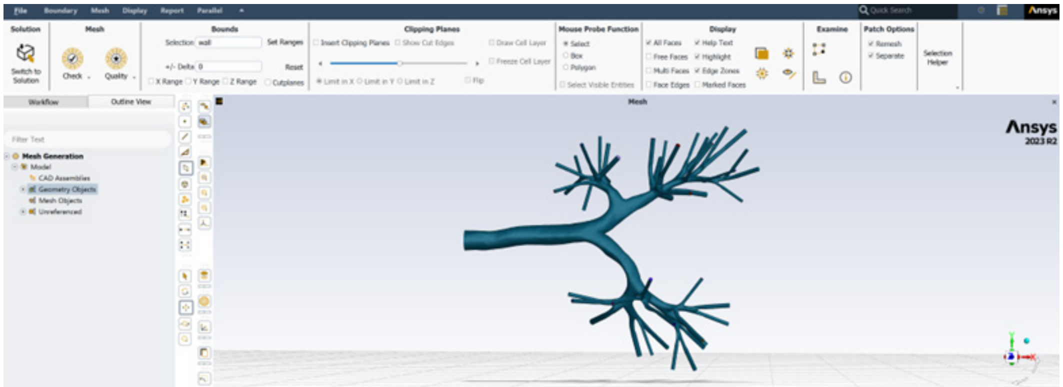


Visualize the geometry: Outline View→Right Click "Geometry Objects"→Click "Draw All"→If needed, click "Fit to Window " (button highlighted by arrow) to visualize the geometry properly.



1 This tutorial was made with the Ansys Fluent 2023 R2 release. Menus may look different, depending on which release is being used.

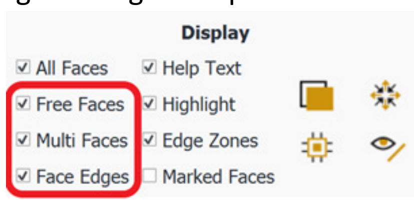
You should now be able to see the geometry similar as shown in the snapshot below:



Note: ASCII format is preferred for the stl file to be imported, which will have the surfaces divided rather than a single surface. Outlets of the geometry (small airway openings) are named as “o**”. Airway wall is named “wall”. Trachea inlet is named “inlet”.

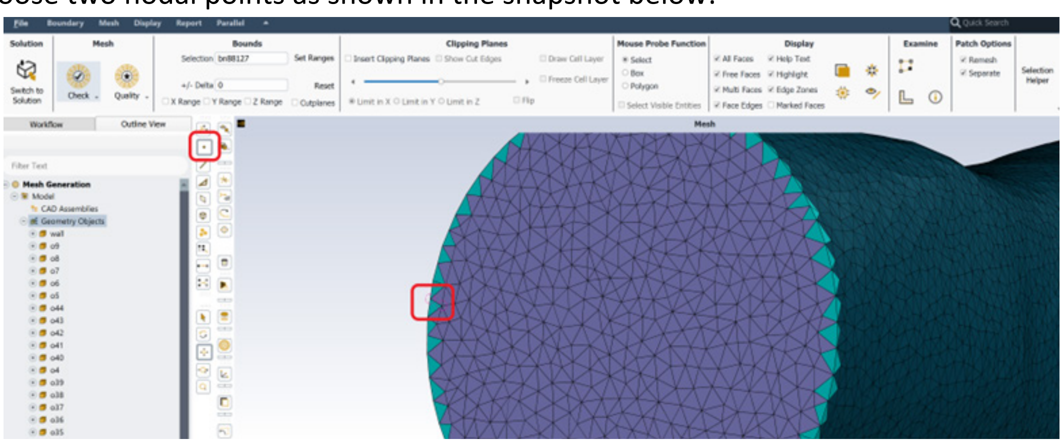
Note: The trachea inlet diameter is 0.016 m in this geometry.

Check “Free faces”, “multi faces”, “face edges” under Display (see snapshot below). This will help to visualize problematic mesh cells during the diagnostic phase of mesh quality later.

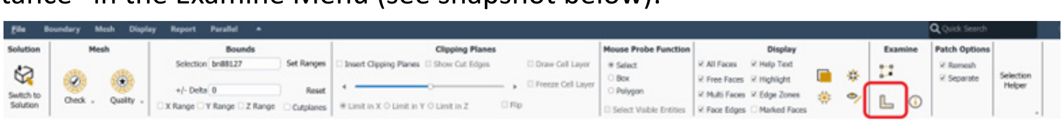


Measure geometric dimensions to determine the proper mesh size for surface mesh:

Click “Node Selection Filter ” →Hold Ctrl on the keyboard and use the left button on the mouse to choose two nodal points as shown in the snapshot below:



Click “Distance” in the Examine Menu (see snapshot below):



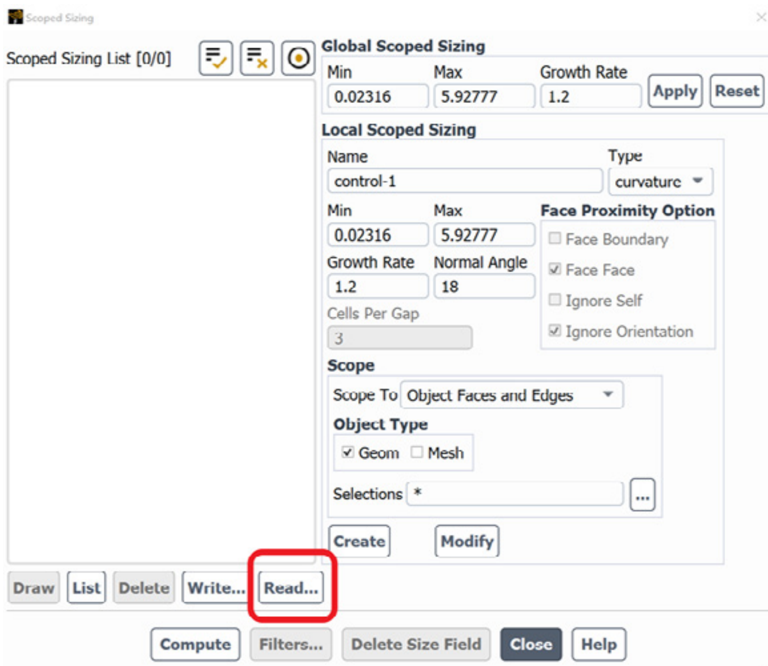
The distance between the two nodes can be found in the Mesh window. It should be approximately around 1.0 in this geometry. It is worth mentioning that this is just a reference geometric dimension for people to use when determining the mesh size. It does not necessarily to be selected in the same way shown here.

After learning the geometric dimension by checking multiple locations, we are ready to set up "Scope Sizing" rules for surface mesh generation.

Note: The recommended Scope Sizing setup below is not the perfect mesh due to the fact that we need to control the total mesh element number below 1 million to make sure the simulation can be run using a free student license[2].

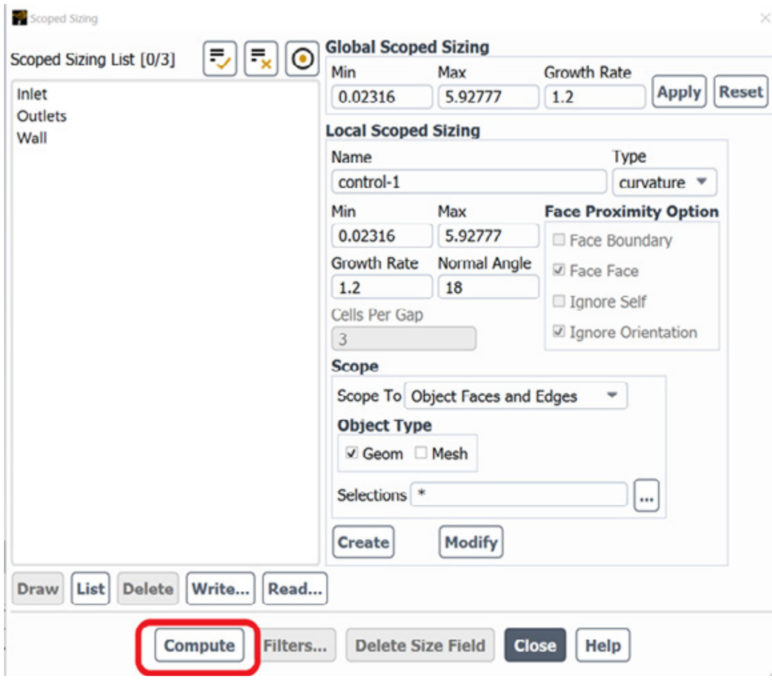
Surface Mesh Generation

Right Click "Model"→ Sizing → Scoped. The window below should pop up for you to set up mesh generation criteria for surface mesh generation.

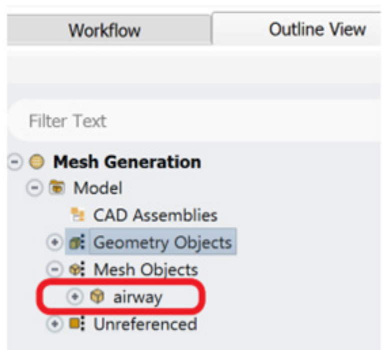


2 Full licenses of the Ansys Fluent software allows for a higher number of mesh elements

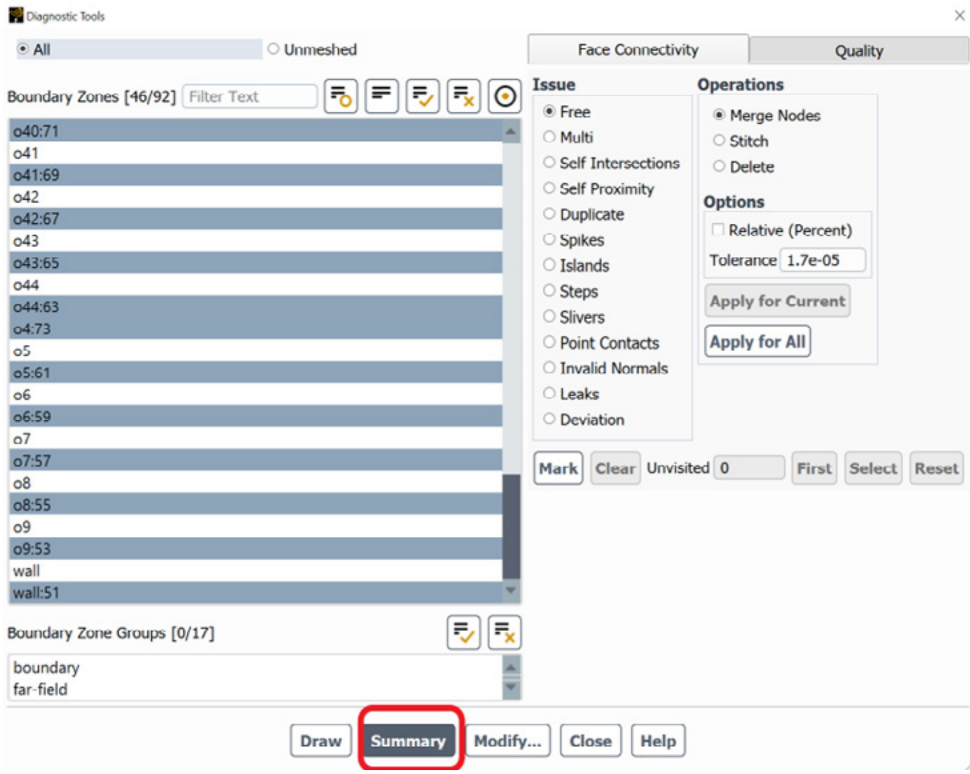
Instead of setting rules manually, please read the scope sizing file named “SurfaceMesh.szcontrol”. Then you should see those scoped sizing rules listed in the window (see the snapshot below). Click “Compute” first, then click “Close”.



We will now create a surface mesh: Outline View → Mesh Generation → Geometry Objects→Hold Shift key and use the mouse to select all surfaces→Right click mouse→Click “Remesh”→Choose “Collectively” →Give a New Object Name “airway”→Click “OK”. The surface mesh will be then generated.



We always need to check the mesh quality and fix the problems before generating the volume mesh: Right-click “airway” under “Mesh Objects”→Diagnostics→Connectivity and Quality. A window will pop up, as shown in the snapshot below. Click “Summary” and check the surface mesh quality report in the console.

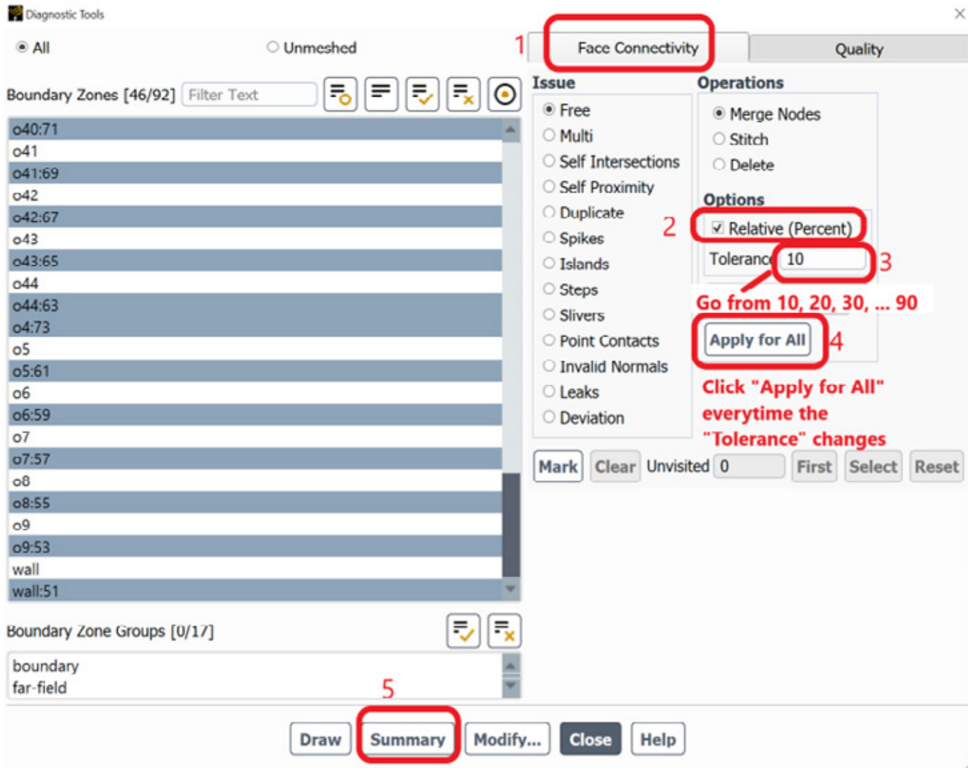


The report should be similar to this:

face-zones-summary	free-faces	multi-faces	duplicate-faces	skewed-faces(> 0.85)	maximum-skewness	all-faces	face-zones
overall-summary	1886	0	0	7	0.94301747	71554	46

Our goal is to make sure: (1) there are “0” free-faces, multi-faces, or duplicate faces, (2) to minimize the maximum skewness as low as possible.

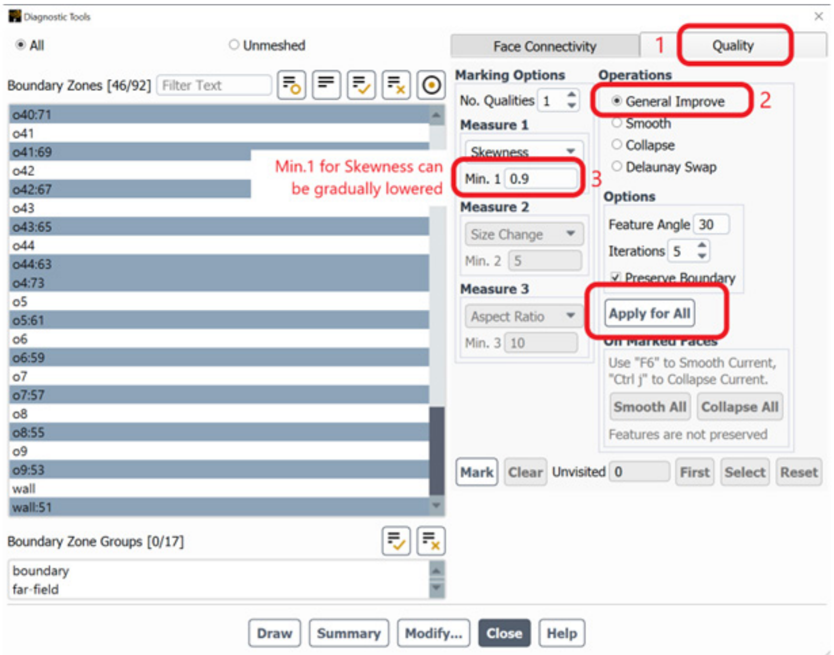
To make the number free faces to zero, follow the steps shown in the snapshot below: Face Connectivity→Check “Relative (Percent)”→Enter 10, 20, 30,…90 once at a time in “Tolerance”→Click “Apply for All” at every time the tolerance is changed.



Click “Summary” to check the number of free faces left and the maximum skewness.

face-zones-summary	free-faces	multi-faces	duplicate-faces	skewed-faces (> 0.85)	maximum-skewness	all-faces	face-zones
overall-summary	0	0	0	7	0.94301747	71554	46

We will now reduce the maximum skewness. Repeat the process shown in the snapshot below. You can always click “Summary” and check the maximum skewness.



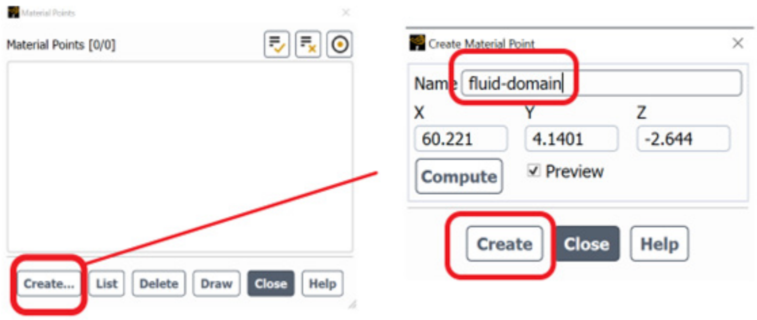
Sometimes, manually fixing the mesh quality is needed to improve the surface mesh. You can click "Mark"→Click "First" to start visualizing the mesh elements with low quality (high skewness). You can manually move nodes or edges of the mesh elements using the tool bar shown below to improve the surface mesh quality. We will not show the detailed process in this tutorial due to the many diversified options you may have.

After reducing the maximum skewness below 0.6 (see snapshot below), we consider the surface mesh quality now acceptable.

face-zones-summary	free-faces	multi-faces	duplicate-faces	skewed-faces (> 0.85)	maximum-skewness	all-faces	face-zones
overall-summary	0	0	0	0	0.59614602	71538	46

Create Material Point

Create a Material Point inside the Computational Domain: Right Click “Model”→Material Points→Click “Create…” →Give the material point name as “fluid-domain”→Type in the coordinate as shown in the snapshot below to X, Y, and Z→Preview and check if the material point is in the computational domain or not→Click “Create”.



Note: (1) The material point is to help Ansys Fluent Meshing to determine which simply connected volume is fluid, solid, or dead zones in the geometry. (2) You can also use the “Node Selection Filter” (icon with a dot in the center) to choose two nodes and use their midpoint as the material point.

Create Volumetric Regions

Create volumetric regions to prepare the volume mesh generation:

Outline View→Mesh Objects→airway→right click “Volumetric Regions” →Click “Compute”→Click “OK”

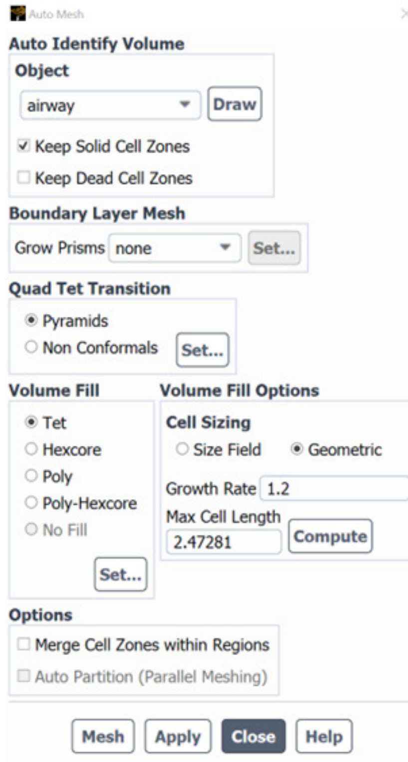
You should be able to see there is a domain generated under “Volumetric Regions”. It is currently named “wall” and identified as a “Solid Domain”. We need to change the name to “fluid-domain” and the type to “Fluid Domain”:

Right click “wall” under “Volumetric Regions →Change Type→Fluid Domain

Right click “wall” under “Volumetric Regions →Manage→Rename→Change the name to “fluid-domain”.

Generate Volume Mesh

Outline View→Model→Mesh Objects→Volumetric Regions→Right click "Cell Zones"→Click "Auto Mesh". A window will pop up (see snapshot below)



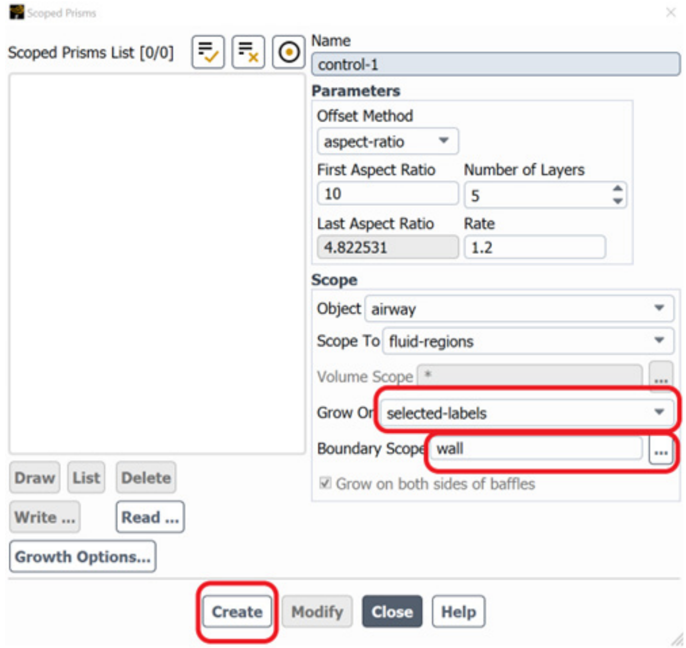
For this tutorial, we will generate a tetrahedral mesh with near-wall prism layers. This window can also generate other types of meshes, but they will not be presented in this tutorial.

In the Window: Uncheck "Keep Solid Cell Zones", then choose "scoped" in Grow Prisms→Click Set

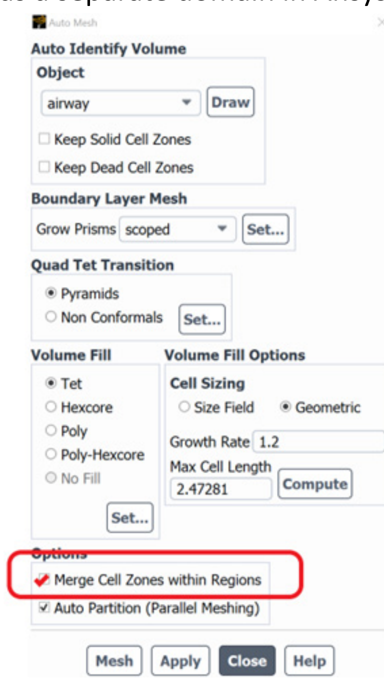
Note: The reason to generate near-wall prism layers is to control y+ to make sure that the near-wall mesh meets the requirement of different turbulence models (e.g., Transition SST) if transitional flow or turbulence needs to be modeled (Feng et al., 2021).

Follow the setup shown in the snapshot below and click "Create"→Click "Close". The prism layers will be only grown on walls, not necessarily for inlet and outlets.

Note: You can also read the scoped sizing file named "NearWallPrismLayers.pzmcontrol" for the near-wall prism layer setup.

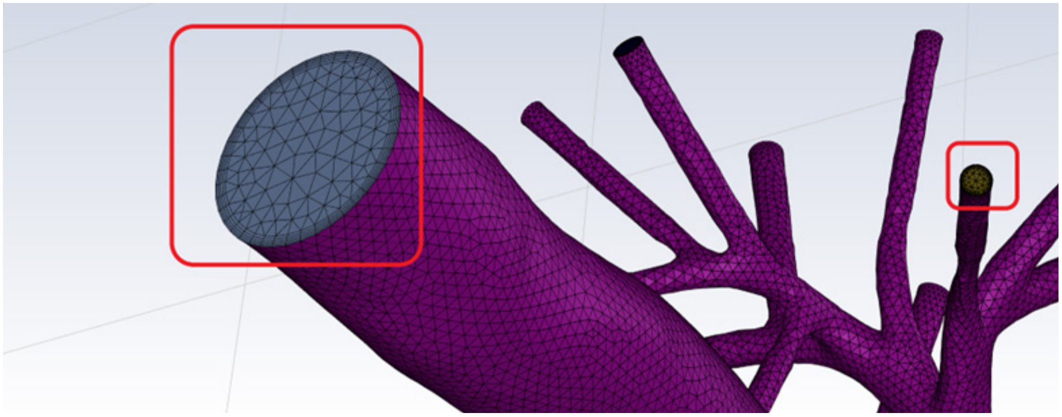


Back to the "Auto Mesh Window"→Make sure to check "Merge Cell Zones within Regions", unless you prefer to have the prism-layer region as a separate domain in Ansys Fluent Solver.

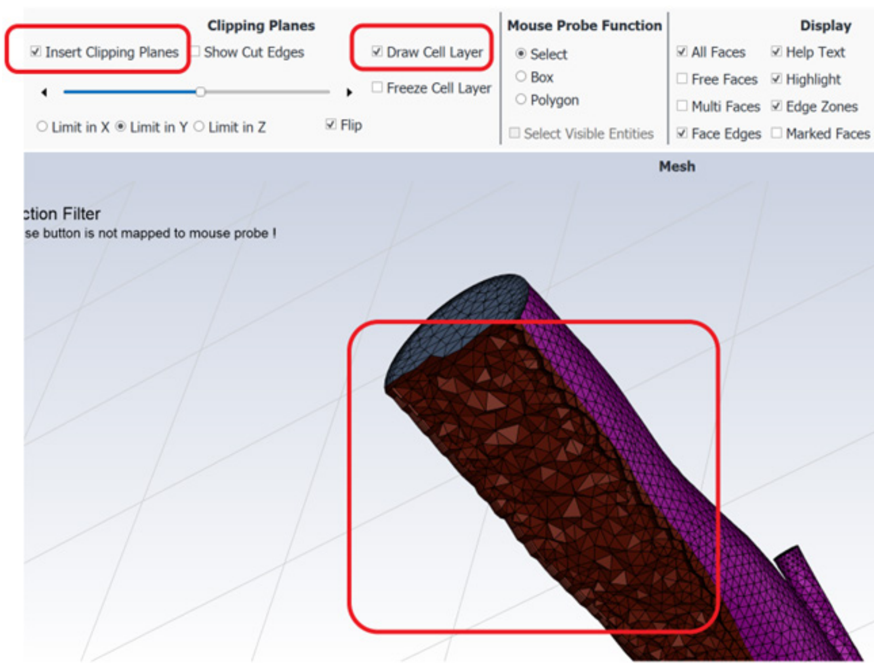


In "Auto Mesh" Window →Click "Apply" →Click "Mesh" to generate volume mesh.

You should then have the volume mesh generated and you may see that the near-wall prism layers were generated by visualize the mesh again using Mesh Generation→Mesh Objects→Right click "Cell Zones" → "Draw All Boundaries". (see the snapshot below)



You can also check the volume mesh by enabling “Insert Clipping Planes” and “Draw Cell Layer” (see the snapshot below).



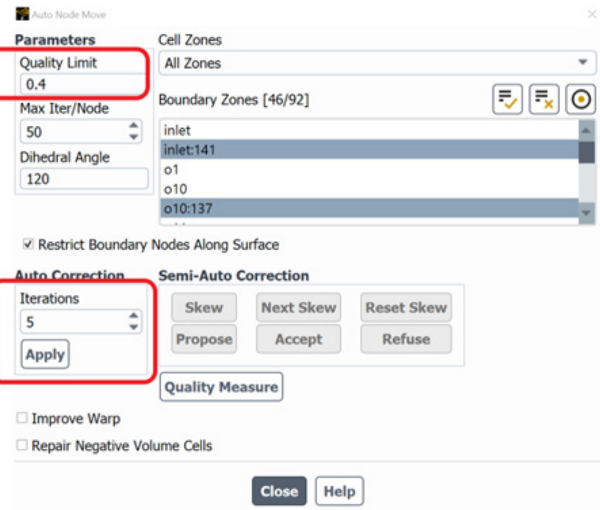
Check volume mesh quality:

Outline View→Model→Mesh Objects→Right Click “Cell Zones”→Summary

Note: The goal is to check the minimum quality and improve it.

Improve volume mesh quality:

Outline View→Model→Mesh Objects→Right Click "Cell Zones"→Auto Node Move→Play with "Quality Limit" and "Auto Correction" to improve the volume mesh quality.



The mesh quality was improved with a minimum quality equal to 0.2688.

name	id	cells (quality < 0.1)	minimum quality	cell count
fluid-domain	2587	0	0.26877599	561571

name	id	cells (quality < 0.1)	minimum quality	cell count
Overall Summary	none	0	0.26877599	561571

Export the Mesh

File→Write→Mesh. You will have a .msh.h5 file for Ansys Fluent Solver. The mesh should contain approximately 560,000 elements (cells).

Note: The .msh.h5 file can also be read by Ansys Fluent Meshing again with all setups saved.

Note: In this tutorial, we only generated one mesh. However, for research, a mesh Independence test is necessary. Details about how to conduct a mesh independence test can be found in Feng et al (2021).

Reference:

Feng, Y., Zhao, J., Hayati, H., Sperry, T., Yi, H. (2021). Tutorial: Understanding the transport, deposition, and translocation of particles in human respiratory systems using Computational Fluid-Particle Dynamics and Physiologically Based Toxicokinetic models. Journal of Aerosol Science, 151, 105672

Use and Reproduction
The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information
This tutorial segment is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources
To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback
Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com