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

Part 5: Post-Processing using Ansys CFD-Post and Ansys EnSight Tools

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 segment uses Ansys Fluent®, the fluid simulation software and Ansys EnSight™, the simulation data visualization software.

Learning Goals

In this tutorial segment, we will cover:

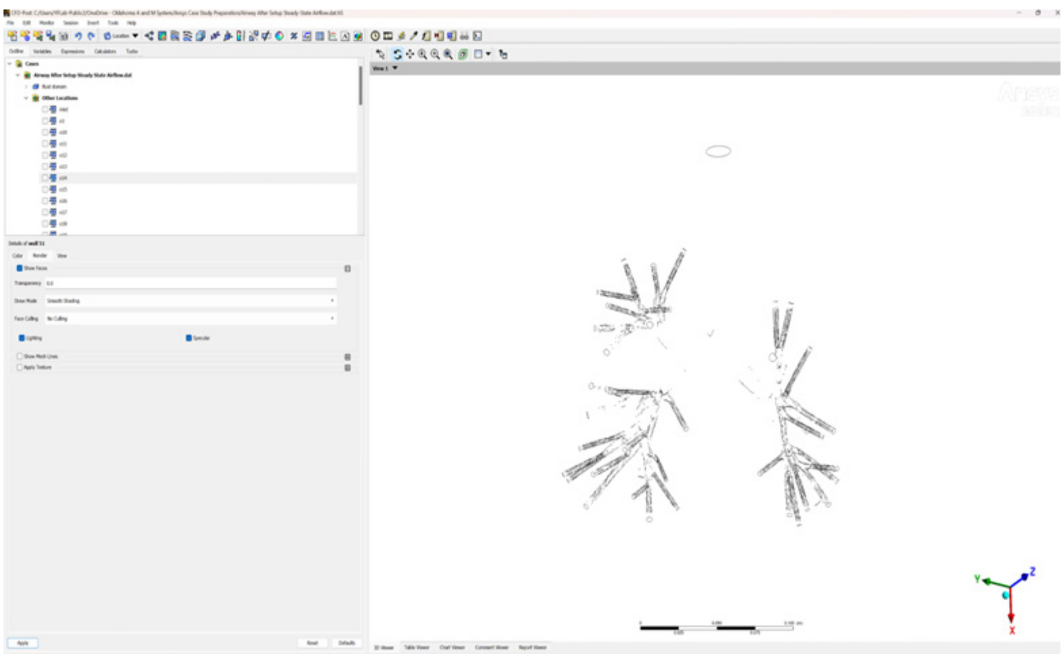
- Contours of spatiotemporal distributions of variables
- 3D distributions of variables using volume rendering
- Particle tracking and residence time analysis.
- Gas concentration profiles.

Steady-State Airflow Simulations

Open Ansys CFD-Post

Load the steady-state airflow simulation result:

File→Load Results→Choose "Airway After Setup Steady-State Airflow.dat.h5"→Open



You should be able to see the geometry appear in the GUI:

Visualize Geometry in CFD-Post

To visualize the geometry in a more professional way to meet the journal paper visualization standard:

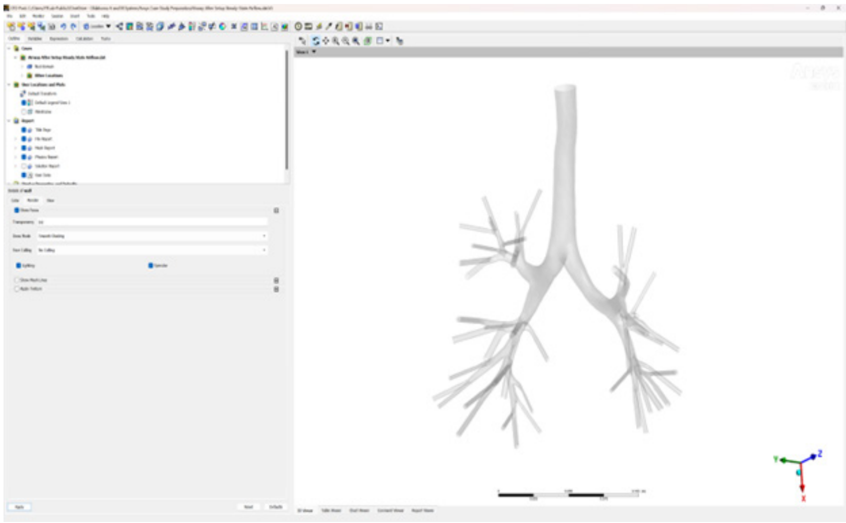
Outline→Airway After Setup Steady State Airflow.dat→Other Locations→Check and Double click "Wall"

Color→change to any color close to gray

Render→Set Transparency to 0.8→Click Apply

Outline→User Locations and Plots→Uncheck "Wireframe"

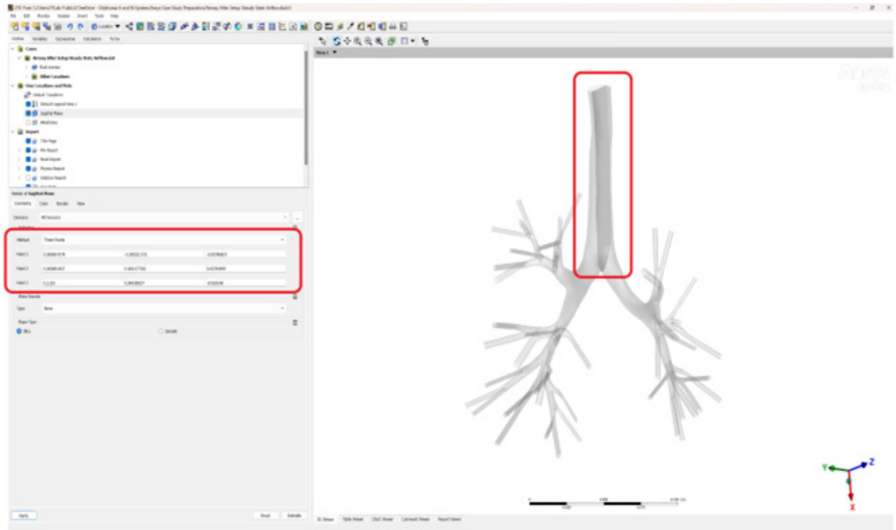
The geometry should look like what is in the snapshot below



Visualize Airflow Velocity Contour

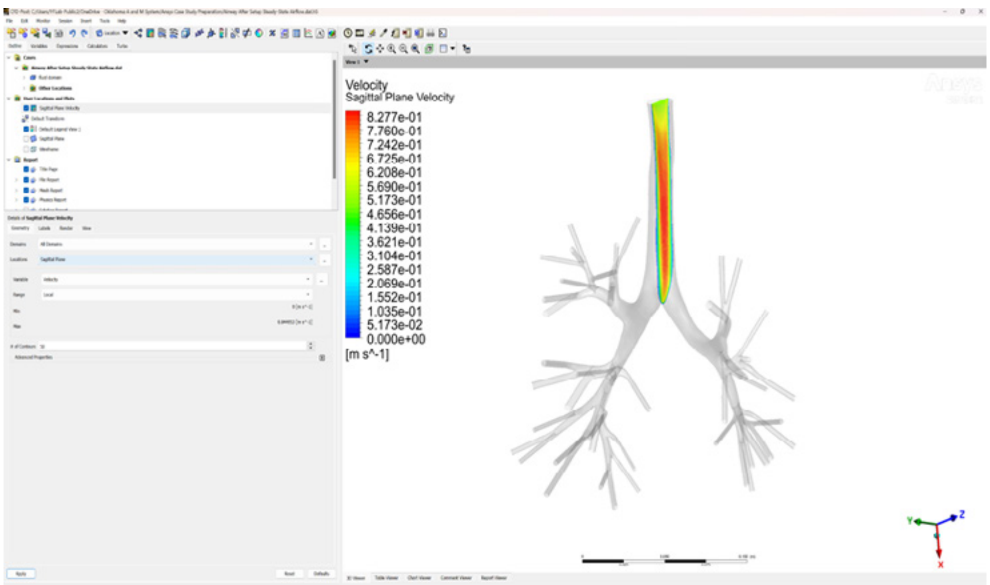
Create the plane: Use the top horizontal tool bar→Location→Plane→Name the plane as "Sagittal Plane"→Click OK→In the menu "Details of Sagittal Plane"→Choose "Three Points" under "Method" →Type in the points coordinates in the snapshot below "Apply."

You should now see a shadowed plane appears in the geometry, which is the current location of the "Sagittal Plane".



Create the velocity contour: Use the top horizontal tool bar→Insert Contour (button with concentric colored circles) →Name "Sagittal Plane Velocity"→click "OK"→Follow the setup shown in the snapshot below and click "apply".

You should now see the velocity contour with a legend in the GUI (see snapshot below):

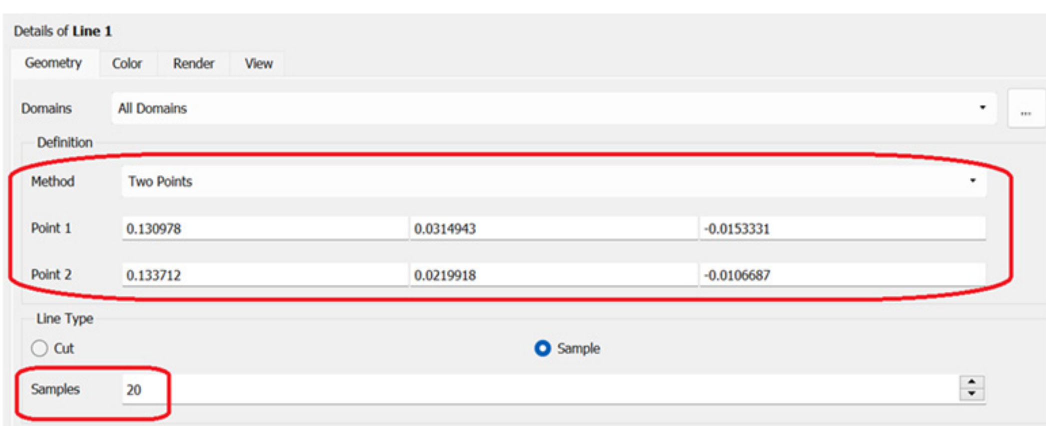


The contour clearly shows how the uniform velocity distribution at the inlet gradually developed and changed to some quasi-parabolic velocity distributions downstream

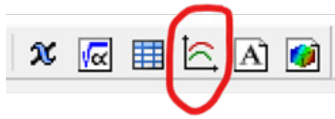
Create Velocity Profiles Along Selected Lines

Create a line: Location→Line→Give a Line Name (e.g., Line 1)→Click "OK"

As an example, please follow the setup in the window shown below→Click "OK".



Create 2D Plot (velocity magnitude vs. z coordinate): Click “Insert Chart” in the toolbar on the top→Name the chart as “Velocity vs Z at Line 1”→Click “OK”.



In the window ‘Details of Velocity vs Z at Line 1”:

General→Choose “XY-Line” in “Type”→Type “Velocity vs Z at Line 1” in “Title”.

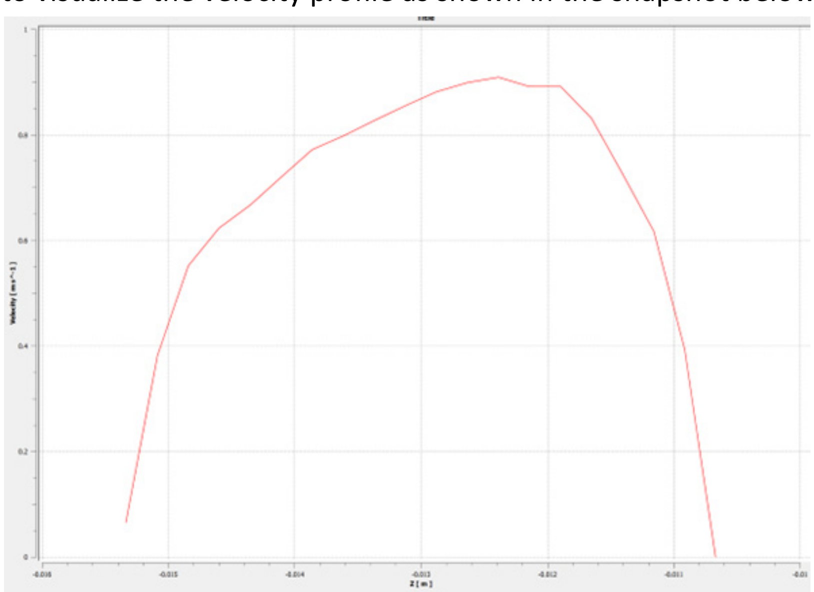
Data Series→Choose “Line 1” in “Location

X Axis→Variable→Choose “Z”

Y Axis→Variable→Choose “Velocity”

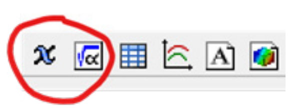
Click “Apply”.

You should be able to visualize the velocity profile as shown in the snapshot below.



You can export the data using “Export” either in .csv format or in .txt format for additional post-processing.

Note: In CFD journal papers for airflow analysis, we usually change Z coordinate into some nondimensionalized variable Z* ranging from 0 to 1. Z* can be created using customized expressions and variables in the toolbar on the top (see snapshot below.



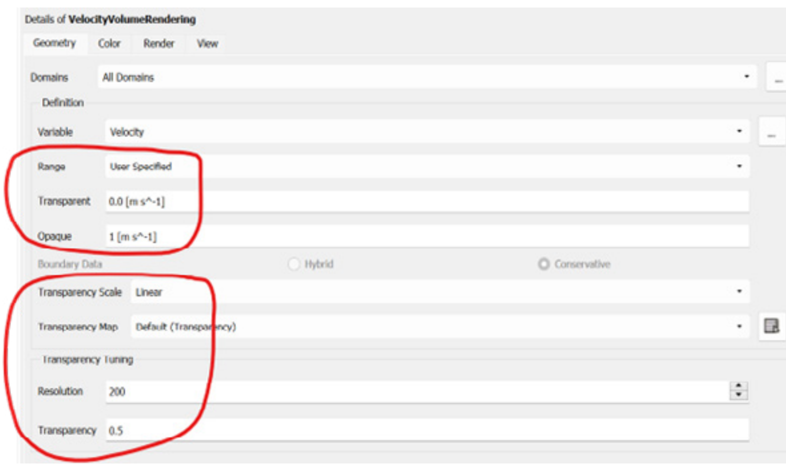
Visualize Velocity Distributions using “volume Rendering”

Volume rendering is an effective method for visualizing velocity distributions in human airways. It provides a comprehensive 3D representation of the flow field, allowing for the observation of complex velocity patterns and interactions throughout the entire airway structure. While velocity contours at different cross-sections offer valuable localized insights, volume rendering enables the visualization of spatial continuity and gradients, giving a more holistic understanding of the airflow dynamics within the respiratory system.

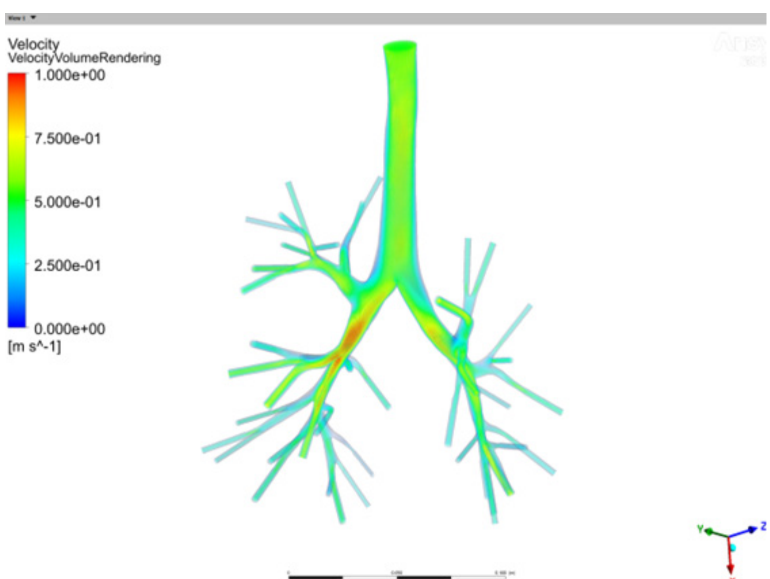
To create a volume rendering visualization:

Click “Volume Rendering” in the horizontal toolbar on top of the screen (see the snapshot below) → Name the volume rendering as “VelocityVolumeRendering”.

Follow the setup shown in the snapshot below →Click “Apply”



The volume rendering will be shown in the snapshot below.



Save .cst File

All post-processing activities can be saved as a state file .cst. You can read the .cst file when loading a new CFD results file, which will automatically conduct all documented post-processing activities saved in the .cst file.

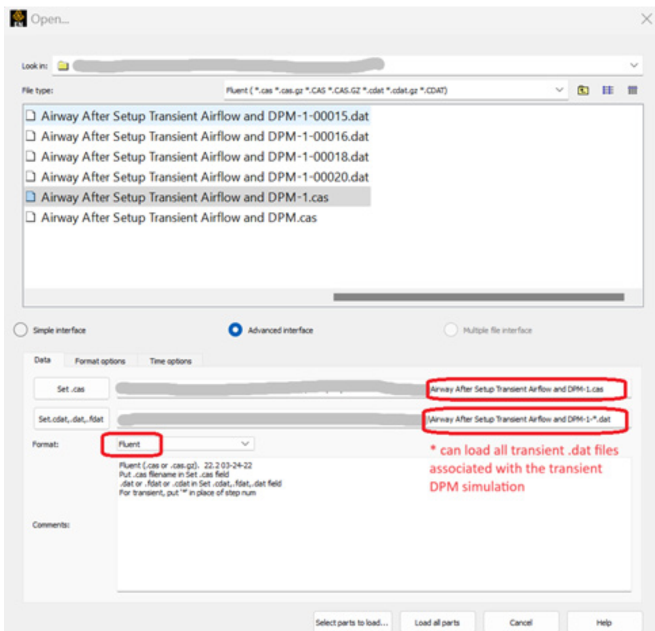
File→Save State

Particle Transport Simulations using Discrete Phase Model (DPM)

To visualize particle transport in “Airway After Setup Transient Airflow and DPM-1-***.dat” associated with the case “Airway After Setup Transient Airflow and DPM-1.cas”, we will use the Ansys EnSight software.

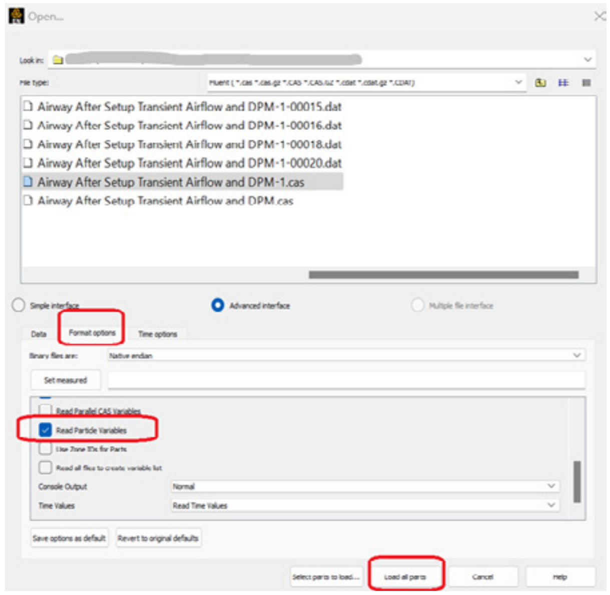
Open Ansys EnSight and Load Transient Ansys Fluent Simulation Data

File→Open→Choose “Fluent “in Format to load .cas or .cas.gz files (see snapshot below)→Select “Airway After Setup Transient Airflow and DPM-1.cas” in “Set .cas”→Select “Airway After Setup Transient Airflow and DPM-1-*.dat” in “Set .cdat,.dat,f.dat”. (see the snapshot below)

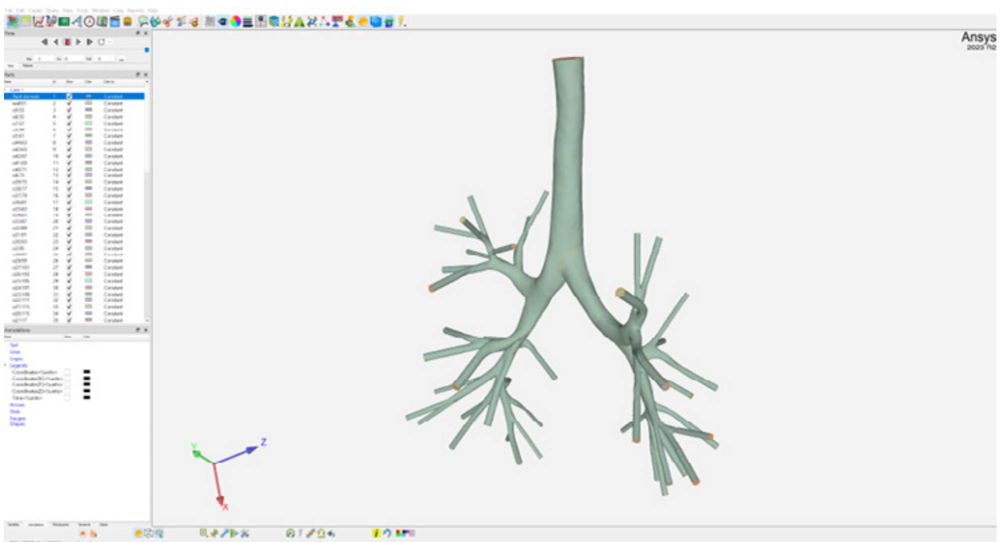


Note: the “*” in .dat file name can enable EnSight to load all transient simulation data files automatically.

In “Format options” tab→Check “Read Particle Variables”→Click “Load all parts”.

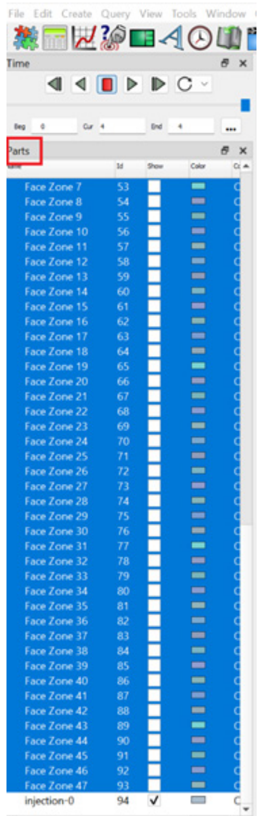


The case and data files will be loaded and visualized.



Visualize Airway Geometry

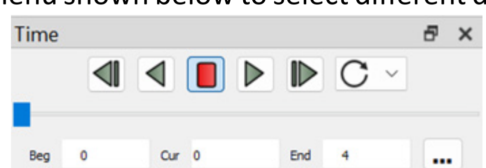
Hide duplicated face zones named “Face Zone *” in Parts menu (see snapshot below): Hold Shift key on the keyboard and use left button of the mouse to choose all faces named “Face Zone *”→Right click the mouse→Click “Hide”.



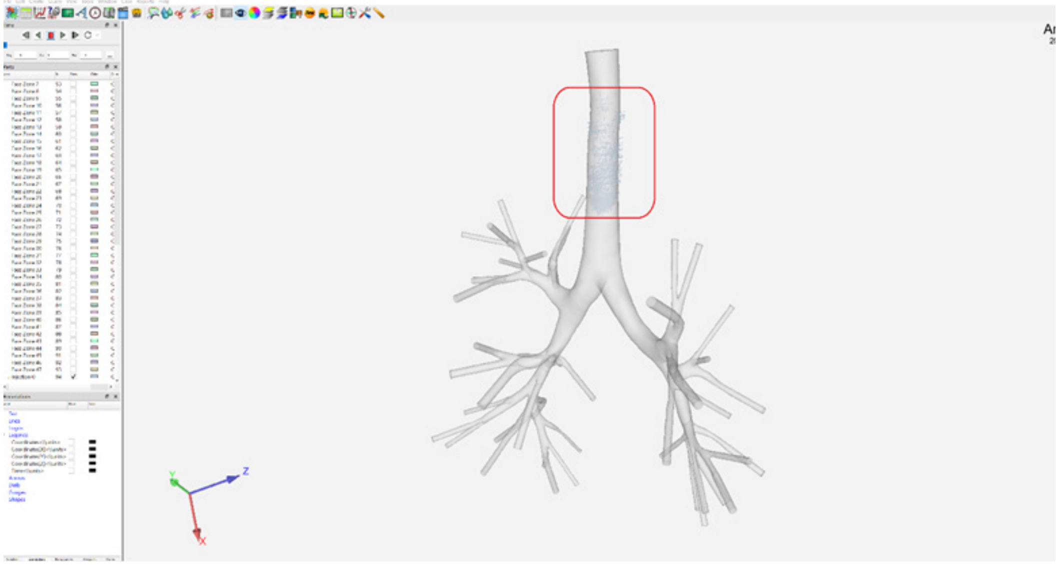
In “Parts”→Choose all visible parts→Right click the mouse→Color→White

In “Parts”→Choose all visible parts→Right click the mouse→Color→Click “Make Transparent”.

Note: You can drag in the Time menu shown below to select different data files at different time station.



You should have a journal paper style transparent airway geometry shown in the snapshot below.

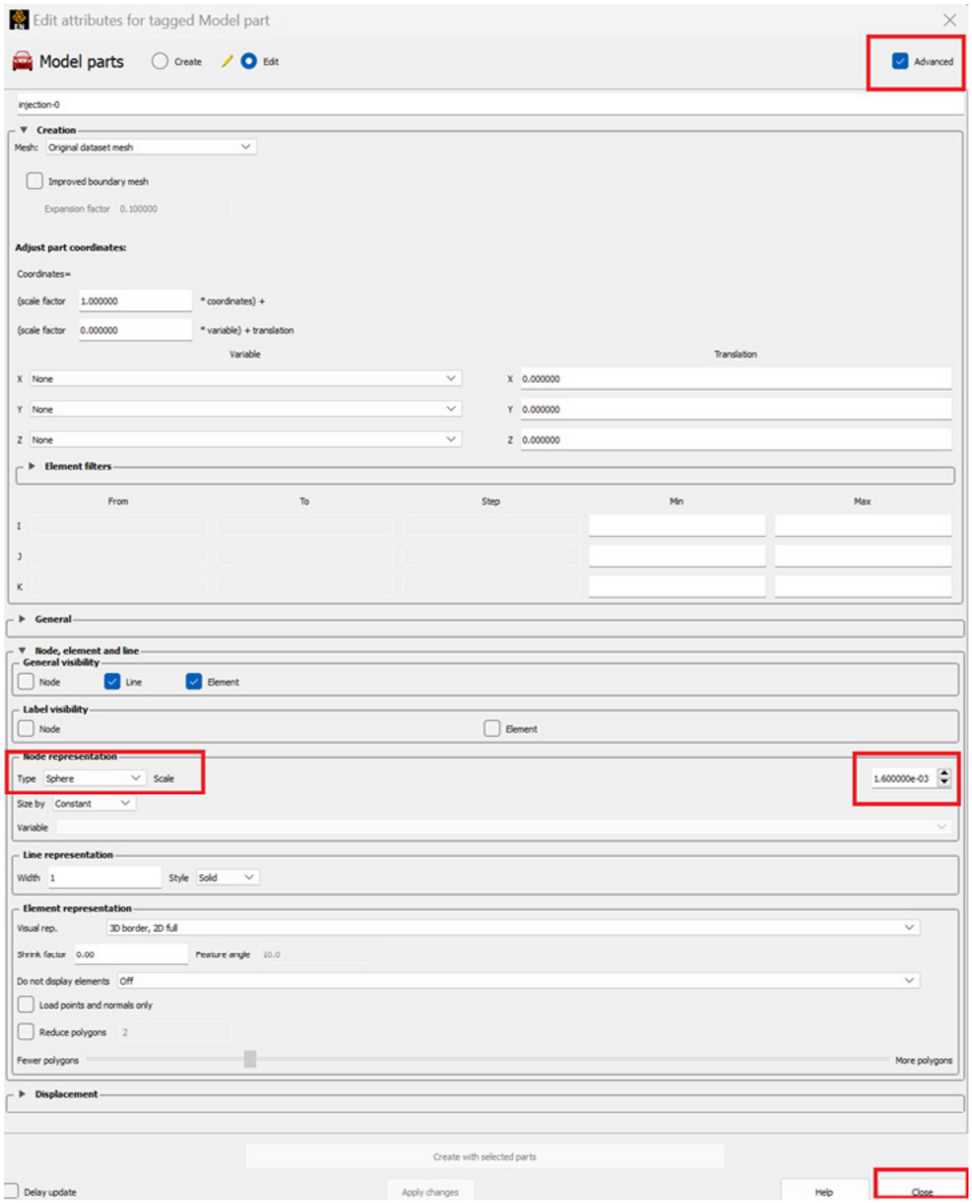


You may see gray cross signs in the trachea. They are the default visualization styles of suspending particles in the computational domain, which is related to the part named “injection-0” in “Parts”.

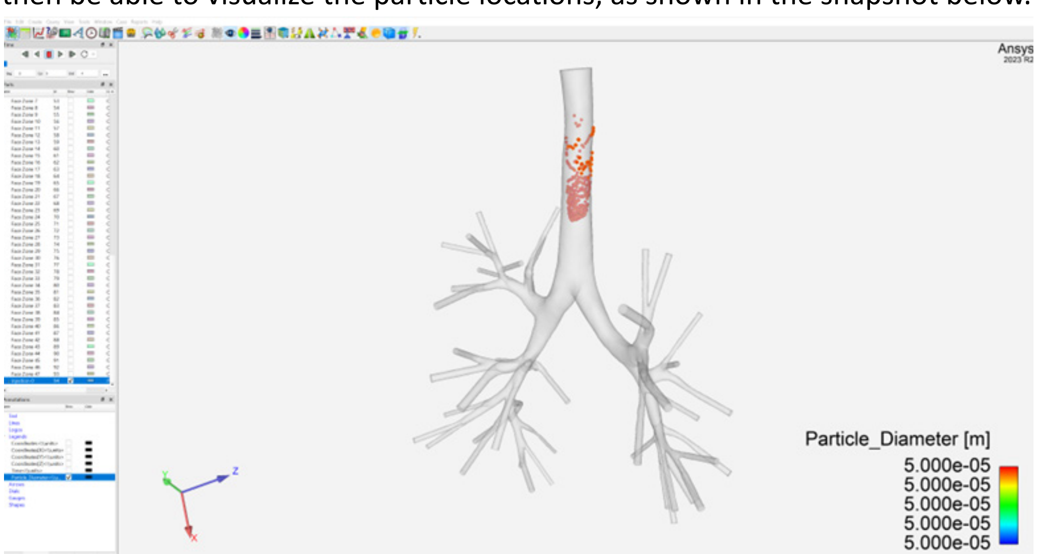
Visualize Suspending Particles

Right click “injection-0”→Color by→Select Variable→Scalars→Particle Diameter→Click “OK”. This step shows the particle colored by its diameter. The diameter legend will be visualized accordingly.

Right-click “injection-0”→Edit→Follow the setup shown in the snapshot below.

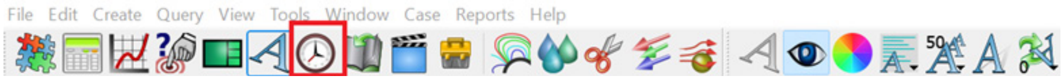


You should then be able to visualize the particle locations, as shown in the snapshot below.

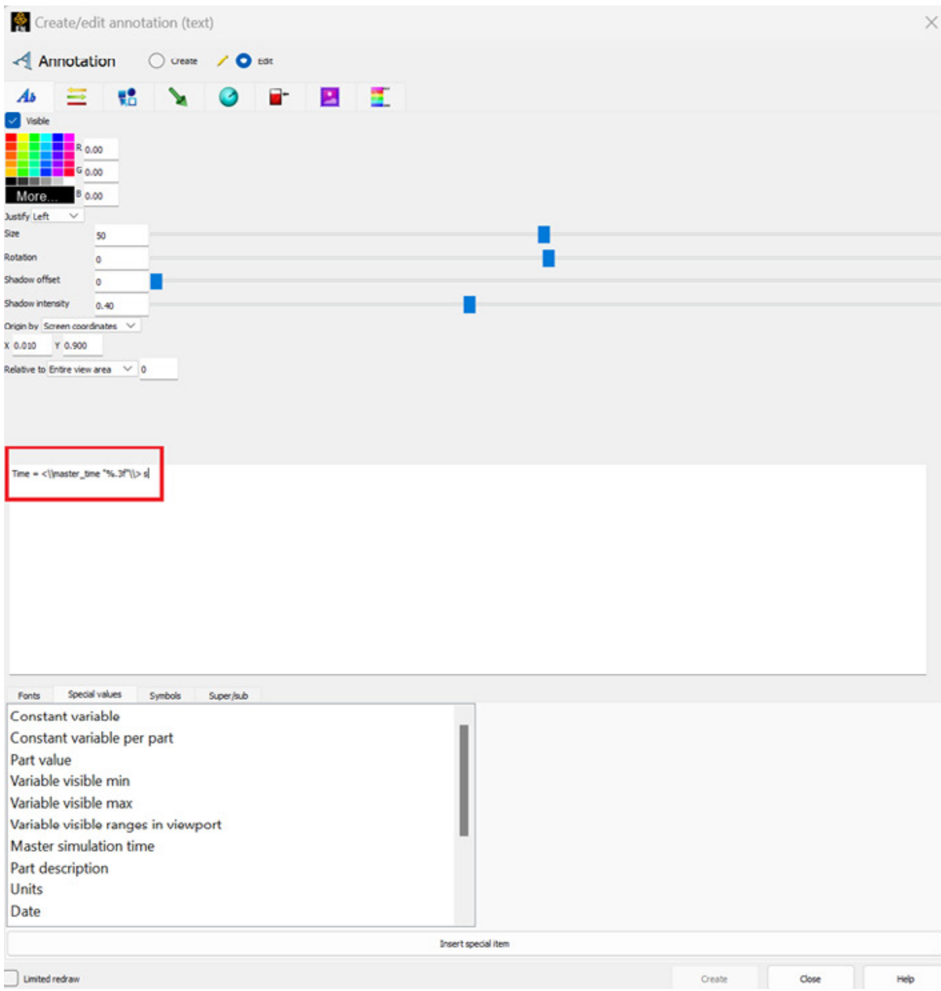


Add Solution Time Annotation

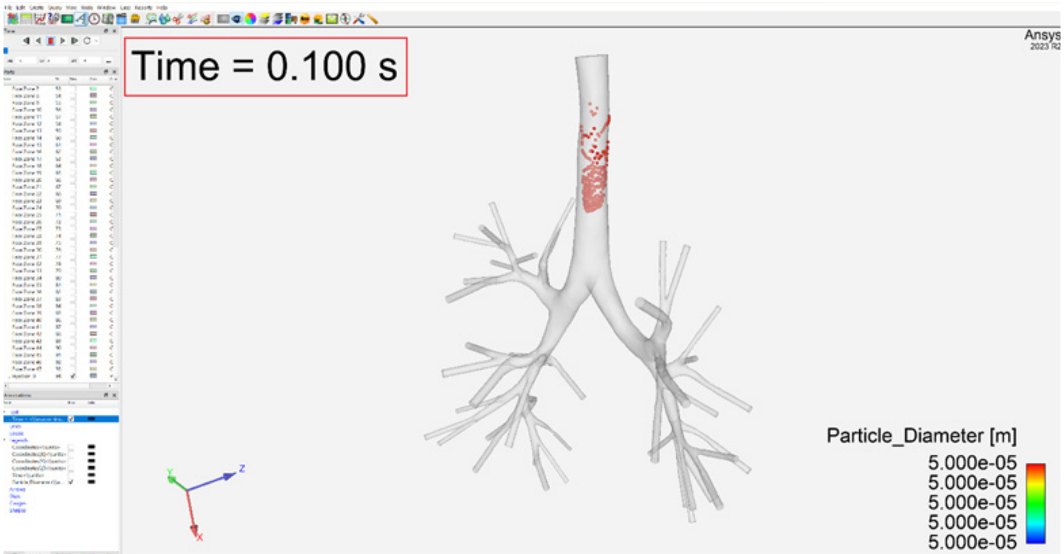
You can also add the solution time: Click “Solution Time” in the toolbar on the top



You can revise the “solution time” annotation using the following step: Right-click “Time” in the Annotations menu at the left bottom of the GUI →Click “Edit”→Change the text “Time = <\\master_time “%.3f”\\>” to “Time = <\\master_time “%.3f”\\ s>” to add the unit of the time “seconds” (see the snapshot below).



The time can be then visualized as follows:



Gas Species Advection and Diffusion Simulation using User-Defined Scalar (UDS) using Ansys CFD-Post

All post-processing techniques described in ‘Part 5: Steady-state Airflow Simulations’ can be applied for UDS visualization. The only modification needed is to change the variable from ‘Velocity’ to ‘Scalar 0’ in CFD-Post. In this session, we will only introduce a couple of tricks for importing multiple transient simulation data files into CFD-Post and discuss some methods for transient post-processing, to avoid duplicated discussion.

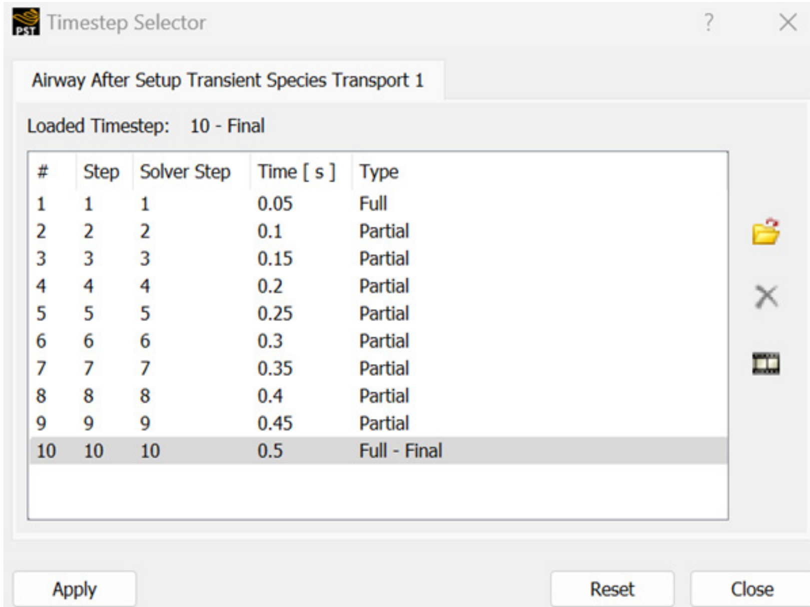
Import Multiple Transient Fluent Simulation Data into CFD-Post

Open Ansys CFD-Post

File→Load Results→Select Transient Fluent Case File “Airway After Setup Transient Species Transport-1.cas”. If all transient simulation data are located in the same folder with the same name before time steps, then CFD-Post will automatically load all simulation data at different time stations.

To check if the transient data at different time stations are loaded or not: Click “Timestep Selector” in the Toolbar on the top (see snapshot below).

A new window should pop up (see snapshot below), in which you can see all data that fully or partially loaded. You can choose which data file you want to work on by clicking the one and then click “Apply”.

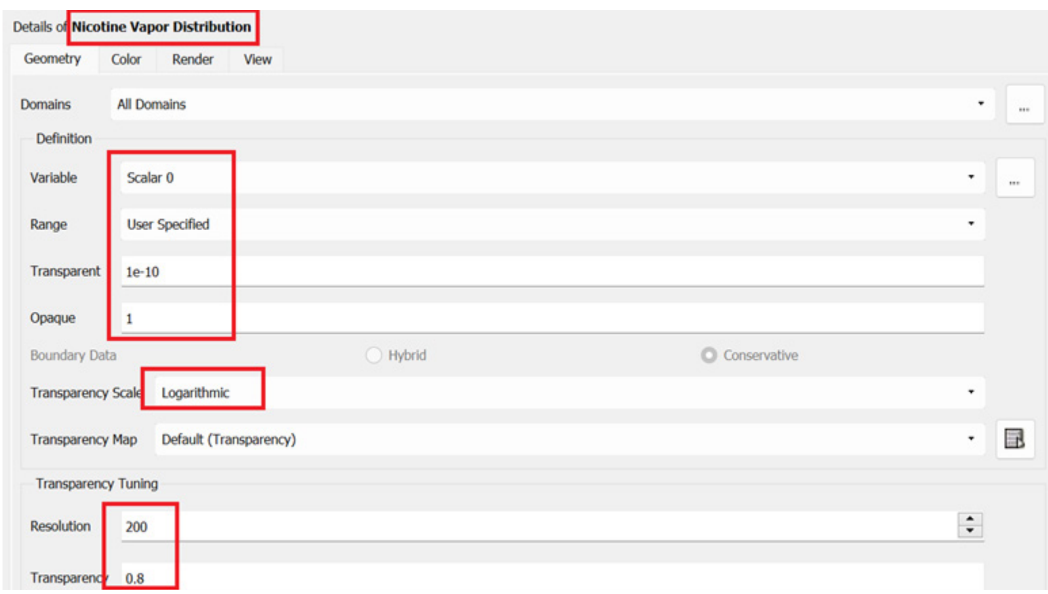


Visualize Nicotine Vapor Concentration using Volume Rendering

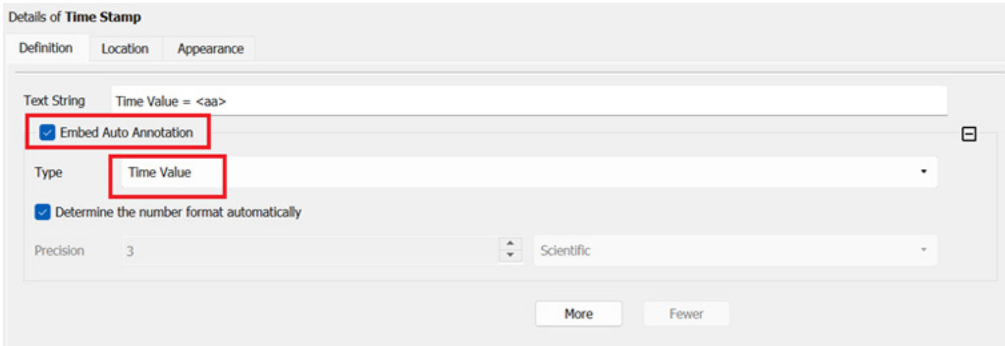
We will load the results at t=0.05 s, t=0.15 s, and t=0.5 s as examples to show how nicotine vapor concentration distributions evolved. We will load t=0.5 s as an demonstration.

Choose #10 result in the "Timestep Selector" window and click "Apply".

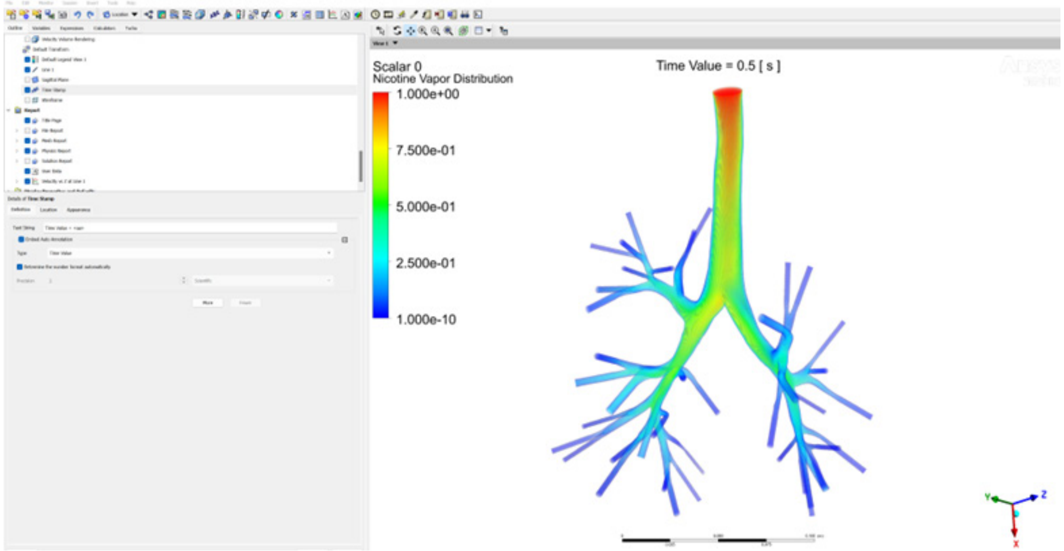
Create a new volume rendering named "Nicotine Vapor Distribution" then setup details as shown in the snapshot below and click "Apply".



You can add a time annotation using: Insert→Text→Give the name to the text as "Time Stamp"→Setup the details of Time Stamp following the snapshot below→Click "Apply"

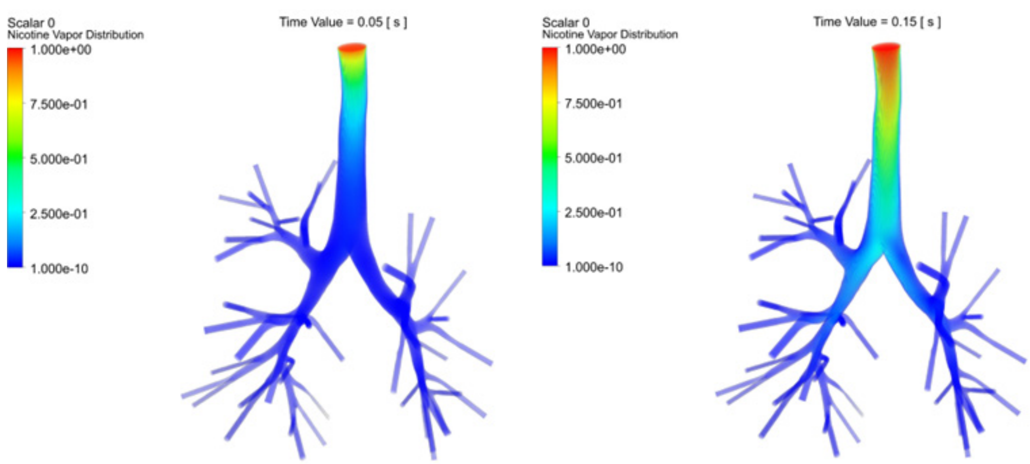


You should now have a volume rendering like the snapshot below.



You can save a new .cst file named “CFDPost-SpeciesTransport.cst”.

You can then load t=0.05 s and t=0.15 s to generate the volume renderings of nicotine vapor concentrations shown like the snapshots below.



Note: As time progresses, the nicotine concentration downstream in the airway increases due to the advection effect. Additionally, it can be observed that the wall concentration of nicotine is zero, which aligns with our wall boundary condition (i.e., Dirichlet B.C.), indicating the rapid absorption of nicotine upon reaching the airway wall.

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 and how to visualize the data.

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