Ansys Software Tutorial

Introduction to Ansys Fluent #3: Steady State vs. Transient Simulation

Mitchell Boots

University of Newcastle

Edited by the Ansys Academic Development Team

education@ansys.com

Summary

Ansys Fluent is a comprehensive computational fluid dynamics (CFD) software that allows you to model fluid domains.

In this set of tutorials, we will introduce basic functionalities of Ansys Fluent through the Ansys Workbench interface. Ansys Workbench is the integration and workflow platform that connects Ansys products.

This tutorial will cover the differences between steady-state and transient simulations, learn how to determine timesteps, and understand how to create videos for visualizing transient flow regions.

This tutorial is #3 of a seven-part tutorial series that serves as an introduction to Ansys Fluent. Details of the topics covered and the order can be found in the table below. These tutorials build on one another, so it is recommended that they are followed in order. Other tutorials can be found on the Ansys Education Resources site.

| Tutorial Order	Tutorial Topic
1	Introduction to Ansys Fluent
2	Mesh Sensitivity
3	Steady State vs. Transient
4	Aerodynamic Analysis Part 1
5	Aerodynamic Analysis Part 2
6	Heat Transfer with Ansys Fluent
7	Compressible Flows

*This tutorial was created using the 2023R1 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the Ansys Learning Forum if you have any questions.

Table of Contents

In this tutorial, we will explore the differences between steady-state and transient simulations, learn how to determine timesteps, and understand how to create videos for visualizing transient flow regions.

We will use the same simulation as we did last week, specifically 'flow over a cylinder,' which clearly demonstrates the distinctions between the two simulation types and allows us to observe the development of flow patterns.
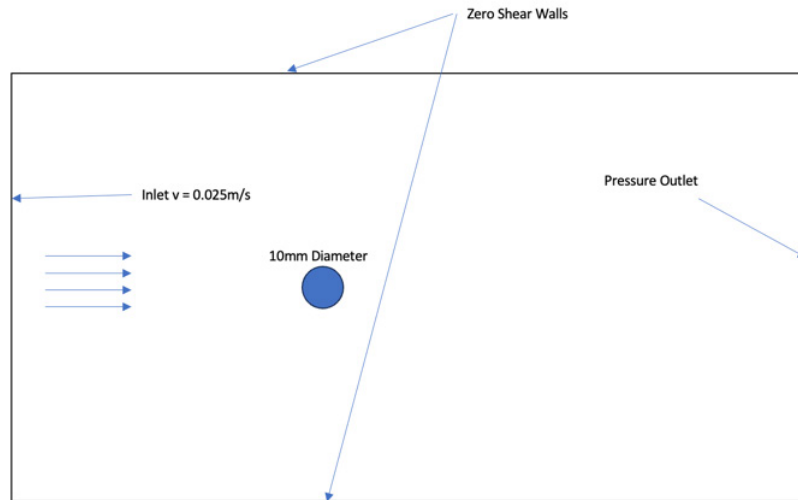
Exercise 1: 2D Laminar Flow over a Cylinder – Steady State:

Utilizing the knowledge you've acquired in the past couple of weeks, create the following geometry and load it into the system.
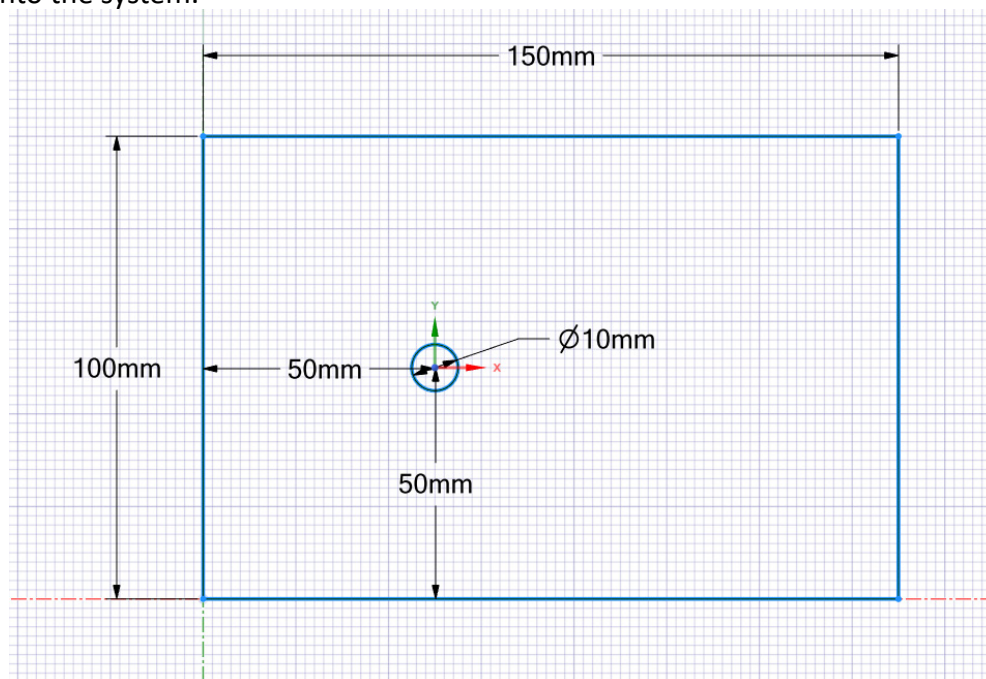

Generate an appropriate mesh for this simulation, utilizing mesh metrics to ensure its adequacy. Additionally, create named selections to identify our boundary conditions: Inlet, Outlet, Walls, and Circle.

Close the 'Mesh' application and launch 'Fluent/Solution'. Complete the following steps to set up the simulation.

1. Select the Laminar Model for the simulation.
2. Assign the 'Fluid' Cell Zone to 'Liquid-Water'.
3. Set the initial velocity to 0.025 m/s.
4. Specify zero shear conditions for the walls.
5. Set the Pressure Outlet boundary condition to 0.
6. Initialize the simulation and run it until convergence is achieved.
7. Compare the obtained results with the reference data.
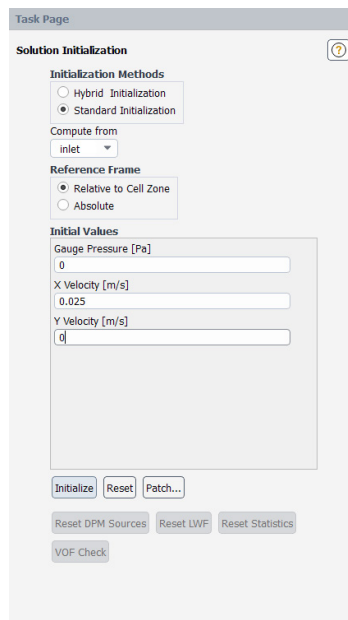
Ansys / FLUENT
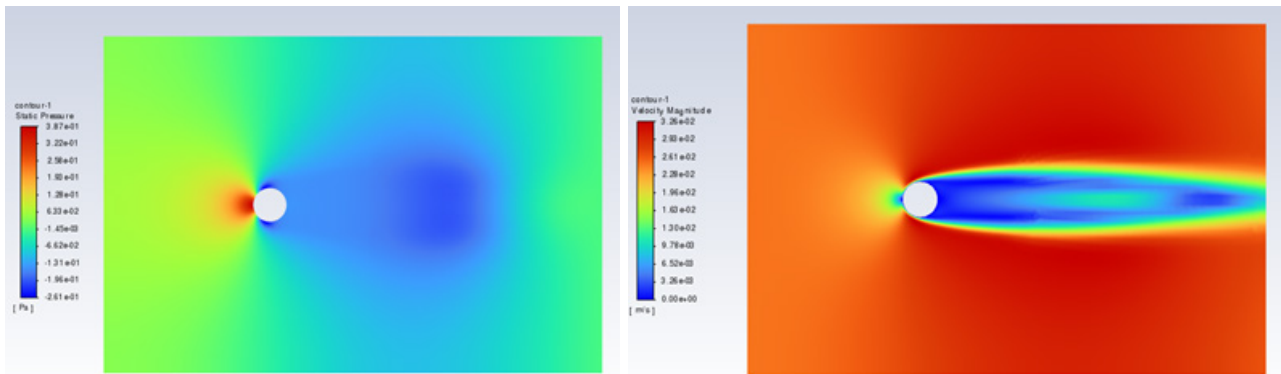
Exercise 2: 2D Laminar Flow over a Cylinder – Transient:

For exercise 2, we will perform a transient analysis on the same simulation. To start, duplicate the first exercise and open Fluent. Modify the simulation by selecting 'Transient' under 'General' settings and check the 'Transient' box.
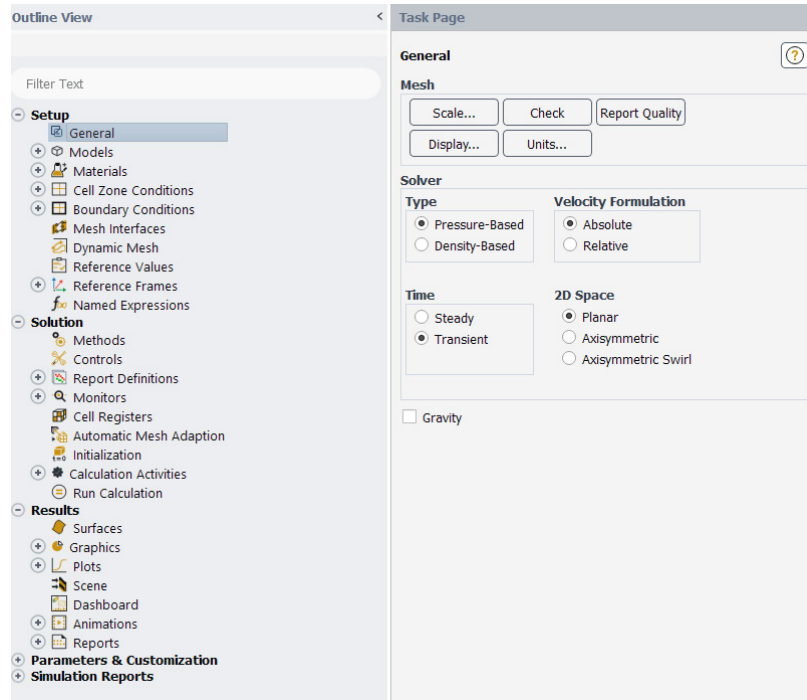
Initialize the simulation again.

Proceed to the "Run Calculation" section. You'll notice a few additional options available. Before progressing further, it's essential to calculate the timestep required. To achieve this, we will utilize the Courant number.

Where:
- Δt is the time step size, representing the small-time interval over which the simulation advances.
- Δx is the spatial grid size or the spatial discretization resolution (max element size).
- v is the characteristic wave velocity or the maximum wave speed in the system.

A commonly used Courant Number is less than 1, and for simulations involving eddies, it's often kept below 0.5. However, you can choose a value that fits your needs. Let's say we select a Courant Number of 0.4.

Example:
Assuming Δx = 0.0025 m.
Assuming v = 0.025 m/s.

Using the Courant Number formula: Courant Number = v * Δt / Δx
So, 0.4 = 0.025 * Δt / 0.0025

Solving for Δt: Δt = 0.0025 * 0.4/0.025 = 0.04 seconds.

Now, let's determine the number of time steps:
You've decided to simulate for 10 seconds.
Number of time steps = Total simulation time / Timestep
Number of time steps = 10 / 0.04 = 250-time steps (approximate).

Finally, let's run the calculation:

Keep the default number of iterations for each time step. Once you've entered the time step size and the number of time steps, initiate the calculation.
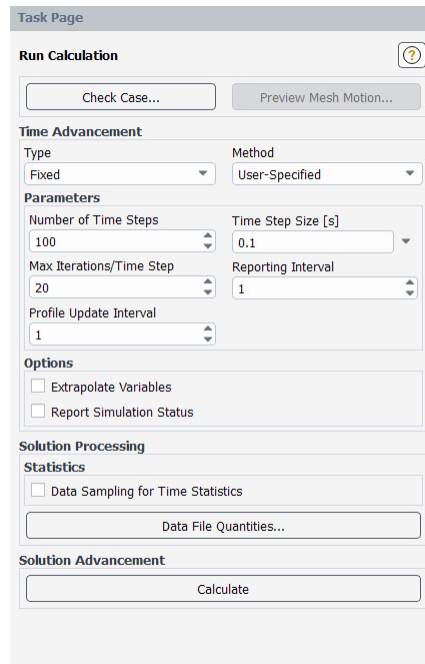
Observe that the loading process and convergence graph exhibit distinct behavior compared to a standard steady-state simulation. This divergence results from iterating each time step for 20 iterations.
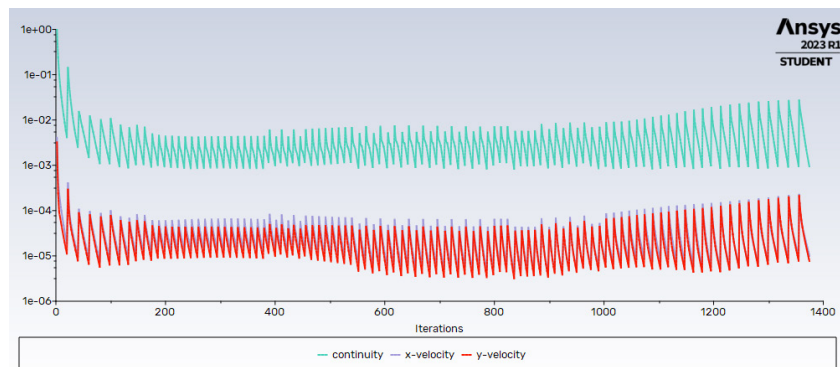

Once the simulation is complete, analyze the pressure and velocity contours. Note that the contour displayed in Fluent corresponds to the final time step. In CFD Post, you have the option to adjust the displayed timestep.

Upon comparison between the transient and steady-state models, you will observe distinct flow patterns that differ considerably.
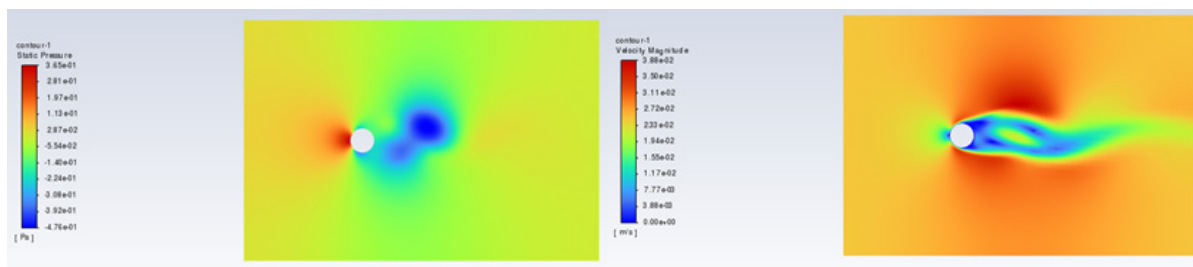
Before closing Fluent I want you to navigate to 'File' -> 'Export' -> 'Case and Data' of the simulation. This will allow us to see the data from each time step in CFD Post.
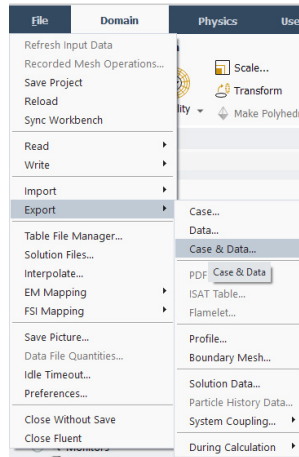

Once the case and data files have been exported, close Fluent and open CFD Post (Results). After the user interface (UI) has loaded, proceed to load the case data into the program.

In the top left corner, select 'File' -> 'Load Results', and then read the file corresponding to the last timestep.
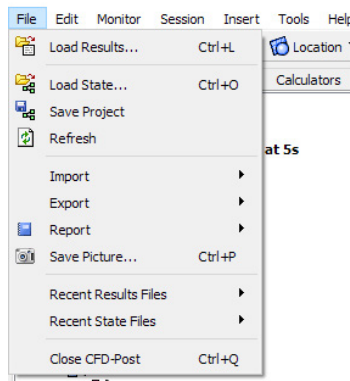

Verify that all the time steps have been successfully loaded by clicking on the clock icon located in the middle of the top row. This action will display a list of all the time steps, organized as follows:
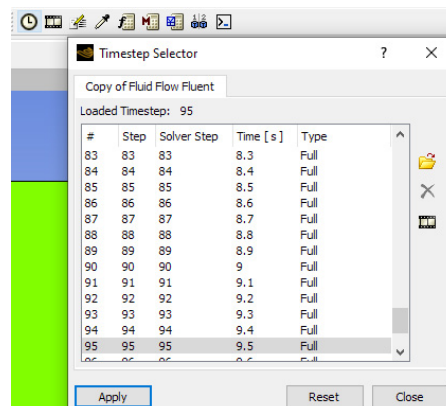
In CFD Post, create a pressure contour on the symmetry 1 boundary. Next, locate the icon resembling a reel of film in the middle of the top row. Click on this icon, and you will be presented with the following prompt:
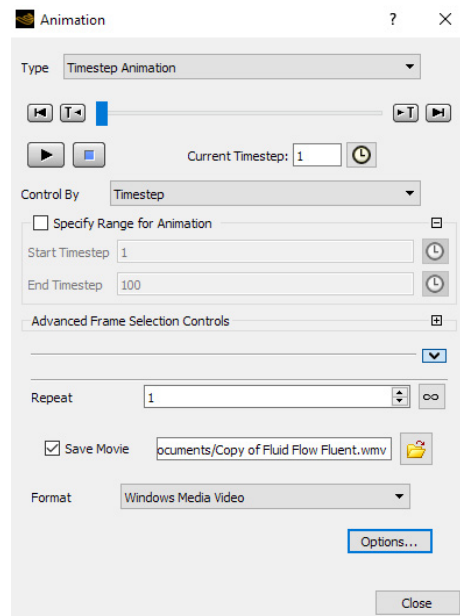

We can generate an animation of the pressure contour by initiating the current timestep at 1 and saving the resulting movie to your preferred location. When you're ready, click the play button. Be aware that cycling through the time steps will require a substantial amount of time.

Once the process is complete, you should find a video at the specified file path location. Watch the video and observe the progression of the flow development.

Repeat the above process, this time focusing on the velocity contour.

Exercise 3: 2D Laminar Flow over a Square Section – Transient:

Now that you've successfully completed your first transient simulation, I'd like you to make a modification to the geometry in SpaceClaim. Adjust the geometry to create a square section with dimensions of 10mm by 10mm.
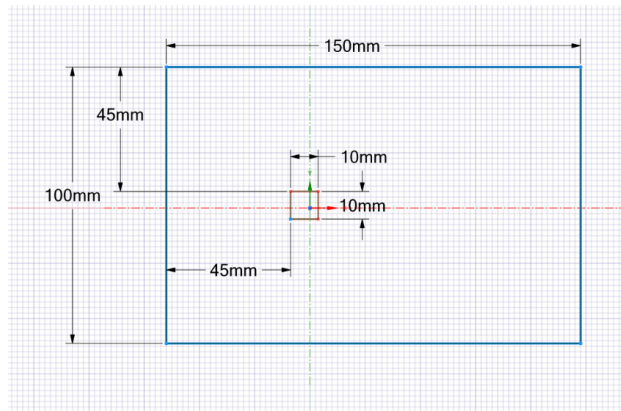
After creating the square section surface, proceed to reconfigure the mesh while maintaining the same named selections.

Navigate to the "Solution" section and apply the identical boundary conditions as in the previous exercise. Run the simulation and then compare the flow patterns between the square and circle geometries.
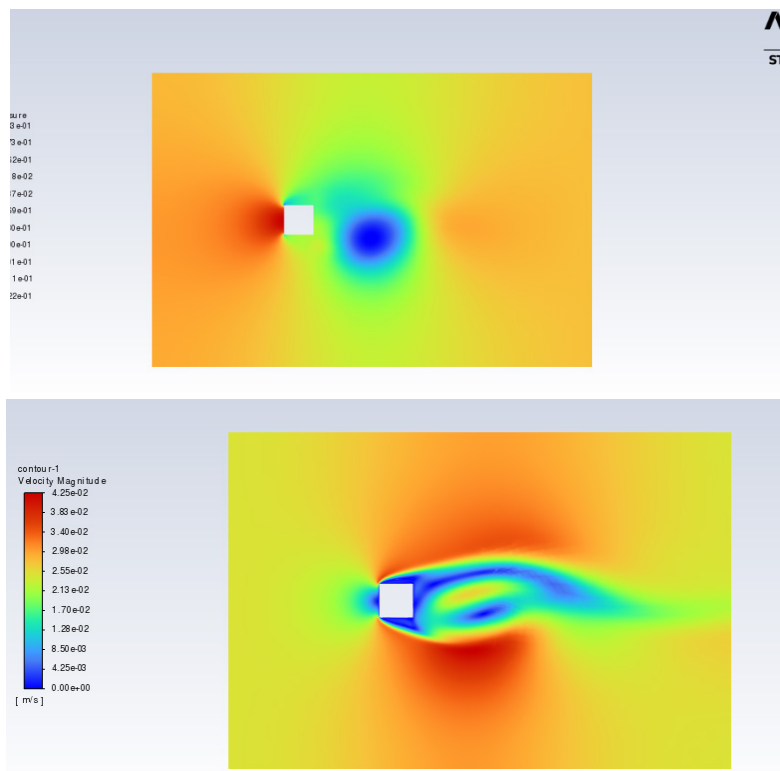
Use and Reproduction

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 case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

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

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit **www.ansys.com** for more information

/ FLUENT