

Getting Started with CFD-Post in ANSYS

Description: An Introduction to Setting Up CFD-Post for Results in ANSYS

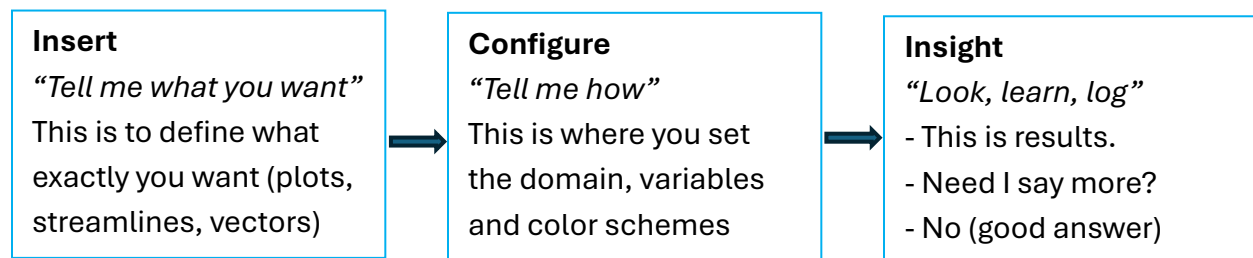
Keywords: Aerodynamics, CFD, Fluent, ANSYS, CFD-Post, CFD Results

So CFD post is a B word!

The UI looks clunky (who designed this UI – such a powerful simulation engine with silly looking user interface, no wonder why mechanical engineers are always raging!). But good news, this is written to simplify some of the aspect of it so that when you start your CFD-post journey, you don't get completely lost.

Lucky you.

But before we start, let us get an overview of the workflow in CFD-post, shall we?

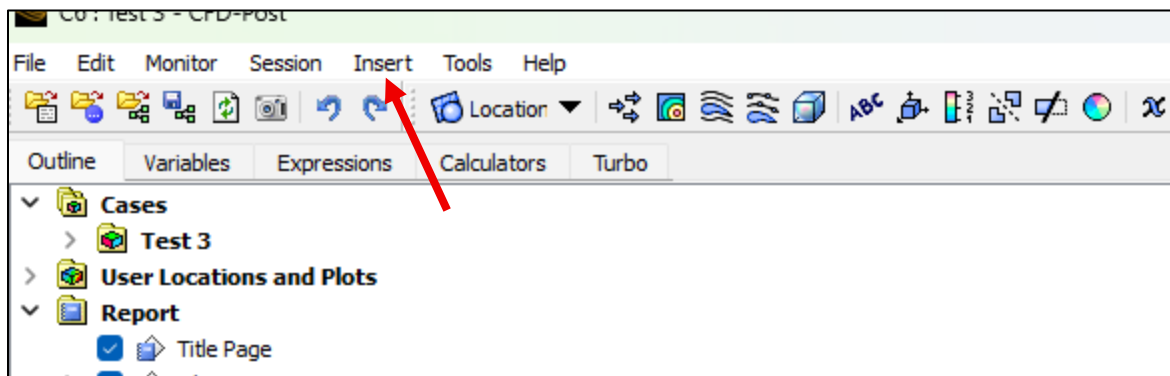


Anyways, let's dig in –

The outline window is basically like the design tree of SolidWorks. If you don't know what that is immediately turn your PC off and sulk in a corner. Come back when sulking is over.

I am not going to talk about that here.

Instead, let's start with the '**Insert**' tab, on the top ribbon of the window.



Insert tab helps you insert various parameters (duh!), from locations like polyline to XY plots, so that you can generate reports and graphs in another platform other than Excel. Go flex on your business major friends.

Here are some options in the insert tab and what they do, in no particular order –

- Vector – for showing direction and magnitude of velocity
- Contour – for visualizing pressure, wall shear stress, velocity etc.
- Polyline (in location) – for creating cutlines (surface tracing, perimeter paths)
- Plane (in location) – (this is needed for a 3D simulation only) for plotting velocity magnitude across a plane, examining pressure-contour on a mid-span cut, extracting XY data across a section
- Streamline – for visualizing fluid paths
- Charts – XY plots (lift/drag breakdowns for example)

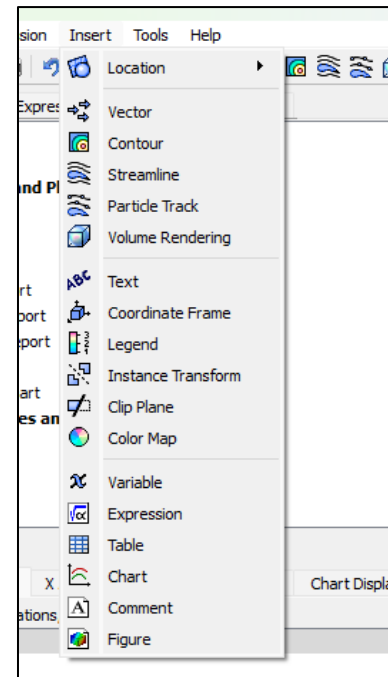


Figure: Insert Tab Shenanigans

For now, that should be enough about insert.

But how do we **‘Configure’** the inserts?

The configuration part is the trickiest part of the whole post processing. You built the mesh like a gigachad, your simulation flew like an aerodynamically efficient wing design, but without specifying exactly what you want to see, those means nothing.

It’s like that quote from *‘The Hitchhikers Guide to the Galaxy’* – Are we asking the right question? Configure to ask ANSYS the ‘right’ question (too philosophical?).

Anyways, that should be that. The configuration part is very case specific as you have already realized. So, I will be discussing them (at length, welcome to the rabbit hole) in the guides.

Now go brew some coffee and go through this again. We don’t want to come back to this when we ask ANSYS the right questions. Right?

Omar Saif

CFD Enthusiast

July 23, 2025