

BEACON Newsletter - September 2024

Introduction to Fluid Dynamics Engineer Role (FMK)

If you are a SOLIDWORKS user and want to run a Computational Fluid Dynamics validation on platform, look no further! Introducing Fluid Dynamics Engineer role aka **FMK**.

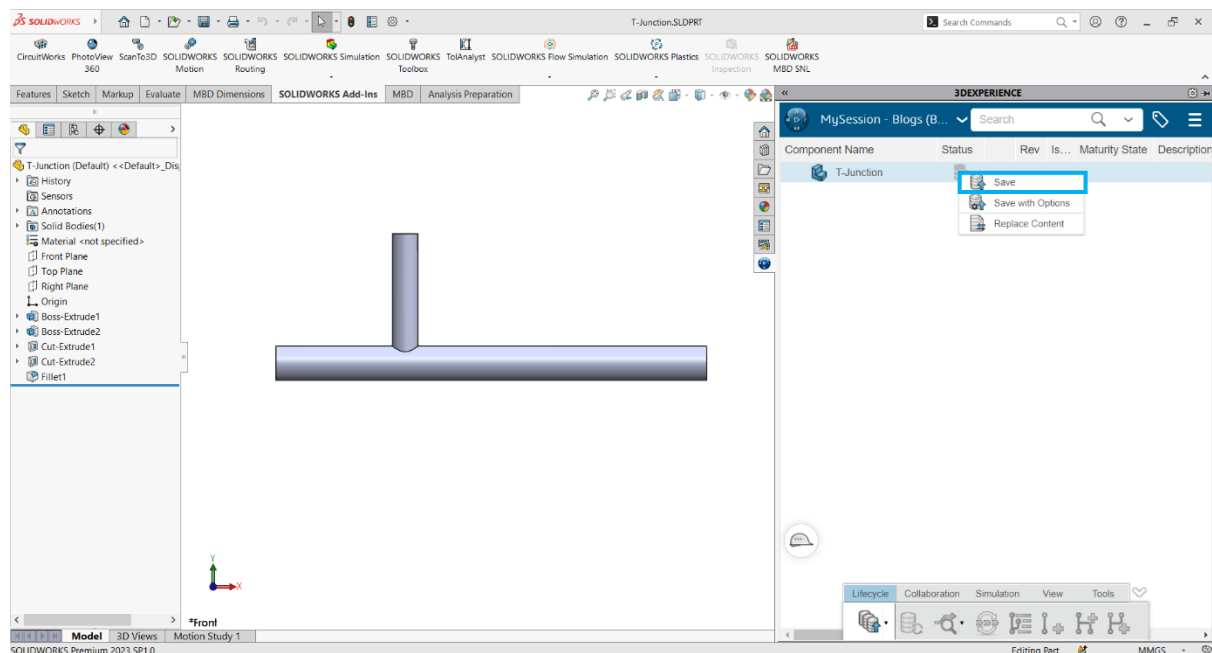
A simulation role for engineers who perform routine fluid flow and conjugate heat transfer calculations to guide design improvements, while exploring innovative design concepts.

With a CAD & PLM-embedded CFD approach, **FMK** enables users to quickly explore 100s of designs while concurrently improving product performance based on criteria such as optimal flow distribution, efficient thermal management, minimal pressure losses, flow uniformity, reduced recirculation etc. **FMK** leverages industry standard RANS based finite volume Computational Fluid Dynamics (CFD) technology with an integrated and fully guided user experience tailored for Designers to perform end-to-end product design efficiently on a single collaborative environment of 3DEXPERIENCE Platform.

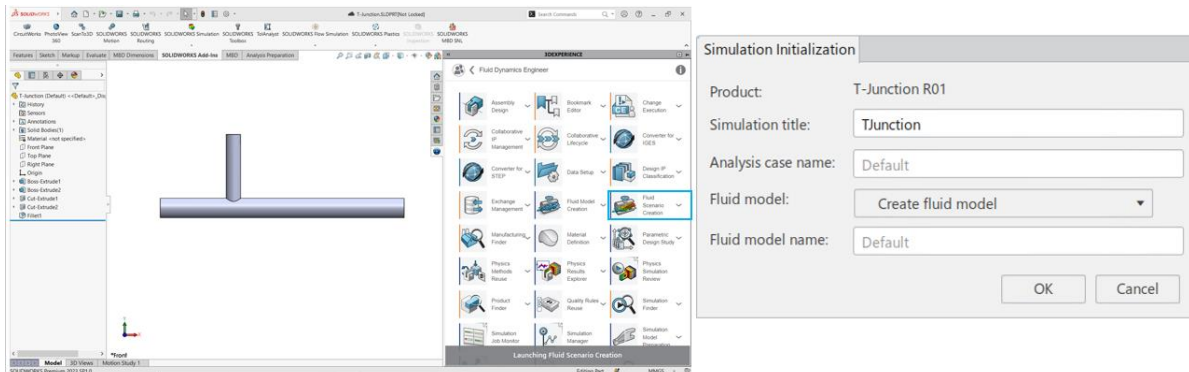
Let's have a look in addressing a T-Junction pipe with fluids entering at two different temperatures.

A **MODSIM** (Modelling and Simulation) approach is used to address the above challenge.

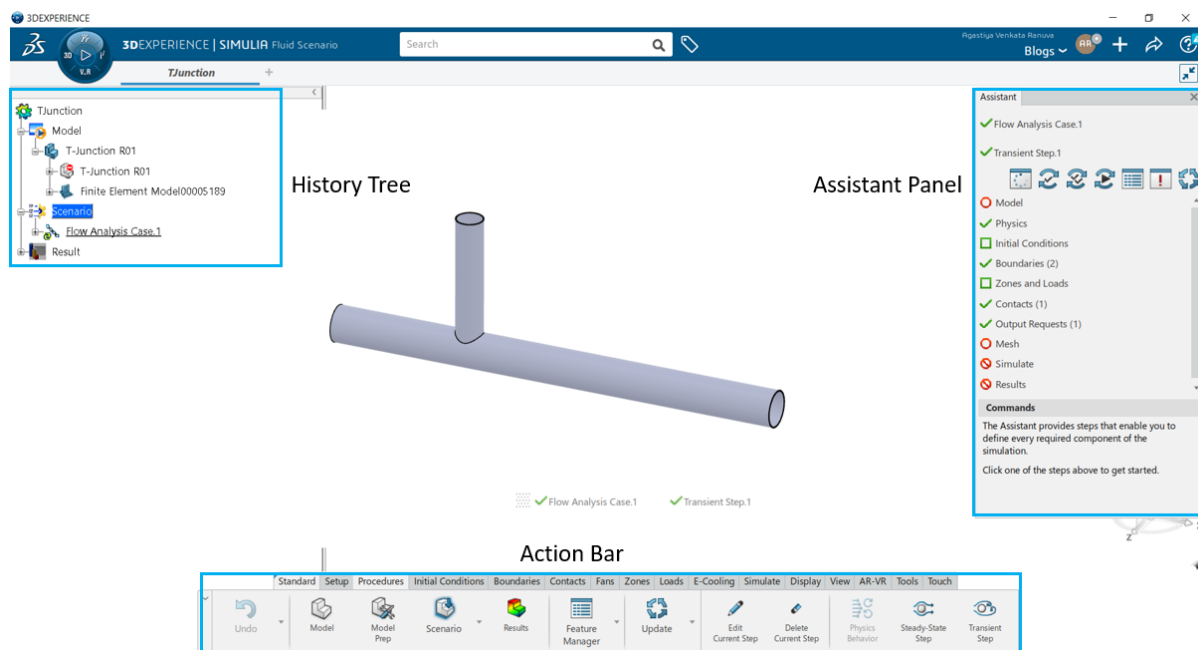
Using the **Collaborative designer for SOLIDWORKS** Role, save the CAD on platform.



Once, we click on save, the CAD file will be uploaded and saved on Platform in selected Collaborative space. Once, the CAD files is saved, the user can launch **FMK** role through compass and select **Fluid Scenario Creation** app. This will launch the CFD Solver on Platform and give a simulation title.



Setup of a simulation study in **FMK**

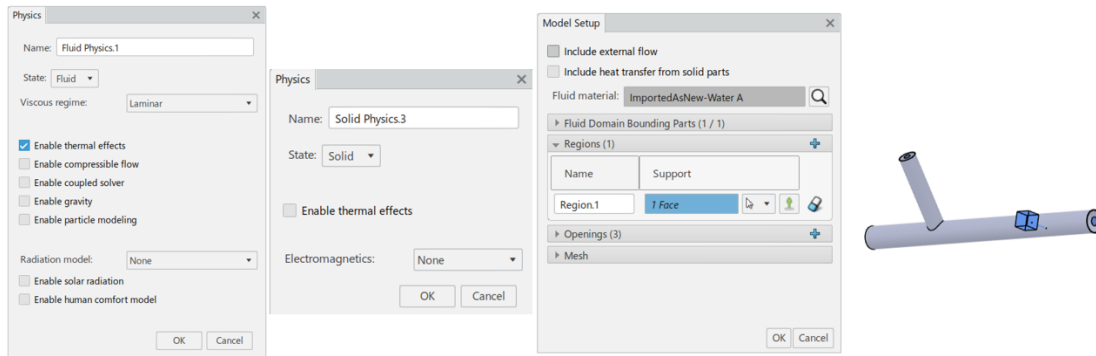


History Tree: Provides the user with all the information that is available and modified during the analysis.

Action Bar: The action bar is the main command bar, commands that are specific to your app are organized in sections in this bar.

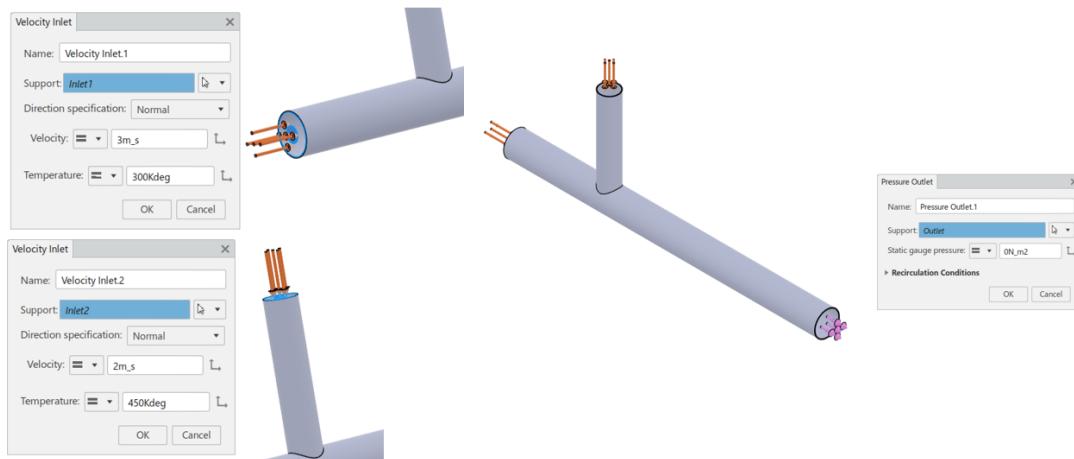
Assistant Panel: The Assistant is a panel that displays a set of actions that you can perform, in roughly the order presented, to complete a simulation.

1. Defining Fluid region:



The first step is to define the fluid physics and region along with the fluid material properties. Since we are using the temperature in the fluids, enable thermal effects and solid physics. The cube indicates the fluid regions and openings can be used to setup the boundary conditions. Also, user has to define whether it's a steady state or transient analysis.

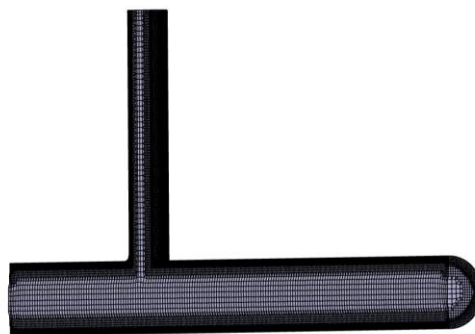
2. Defining Boundary Conditions:



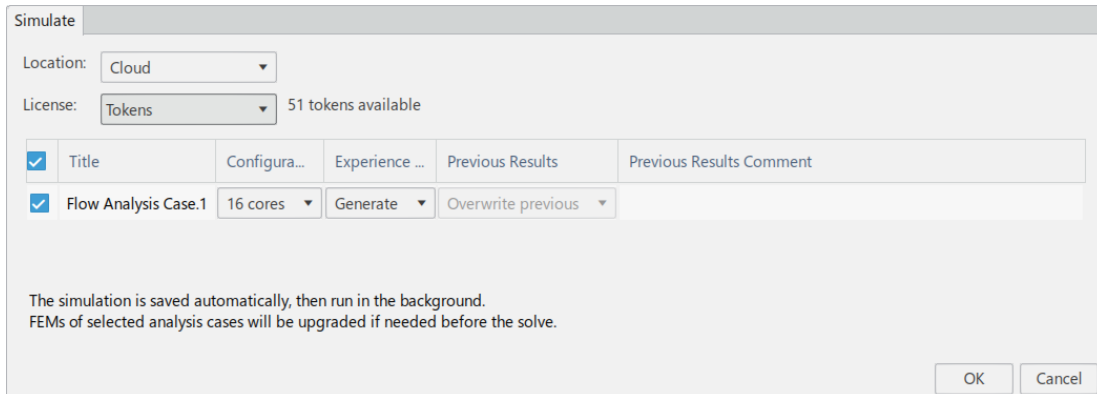
The fluid enters the tube at two different locations with two different velocities and at different temperatures while exiting through the same outlet.

3. Mesh

FMK solver utilizes Body Fitted Mesh.



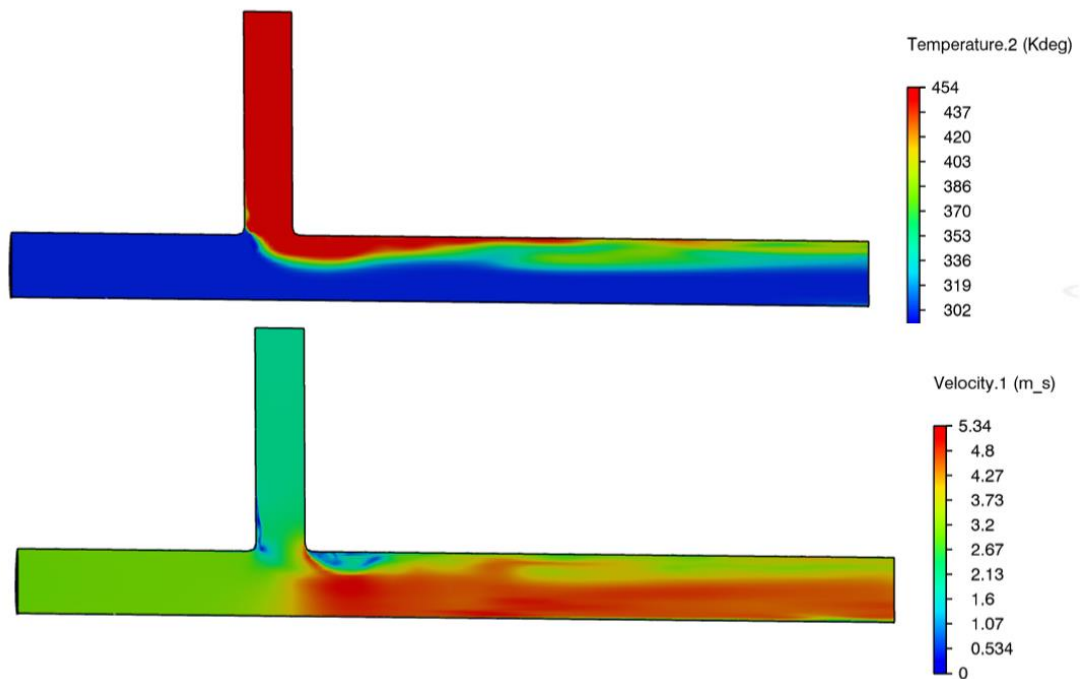
4. Solve

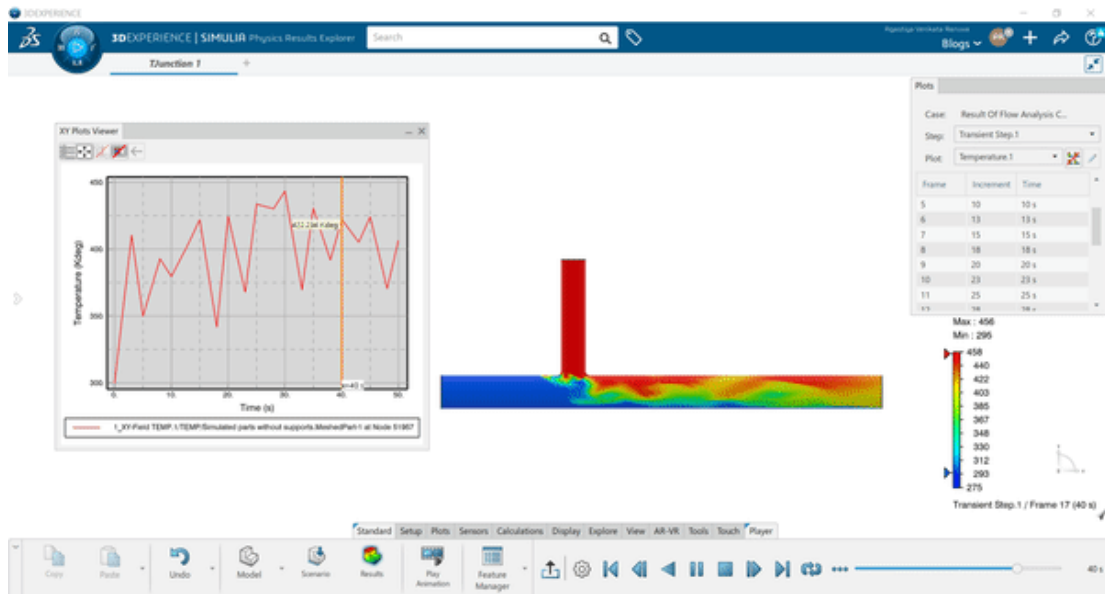


Solver provides the option to execute the simulation in both local i.e., computer and cloud computation. The solver can utilize cores like 16, 36, 72 and 144. But by default, the user can make use of upto 16 cores without any additional costs.

5. Results

Once the analysis is completed, the user can obtain several results like Pressure, Velocity, Temperature, etc., in the form of contours and XY plots (Graphs).





Try our easy to use simulation tools and improve the product performance.

To know more details, please reach out to us at:

Phone: +91 7406663589

Email ID: info@beacon-india.com

Website: <http://beacon-india.com>

Request a Quote Now: <https://www.beacon-india.com/request-a-quote>

Request a Demo Now: <https://www.beacon-india.com/request-a-demo>