

3D Steady Diffusion - Numerical Results

Author(s): Keith Works, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry + Mesh](#)

[3. Model Setup](#)

[4. Numerical Solution](#)

[5. Numerical Results](#)

[6. Verification & Validation](#)

Numerical Results

Now we will look at the results of our simulation. Watch the following video for demonstrations:



The csv file containing the analytical solution is provided on the next page.

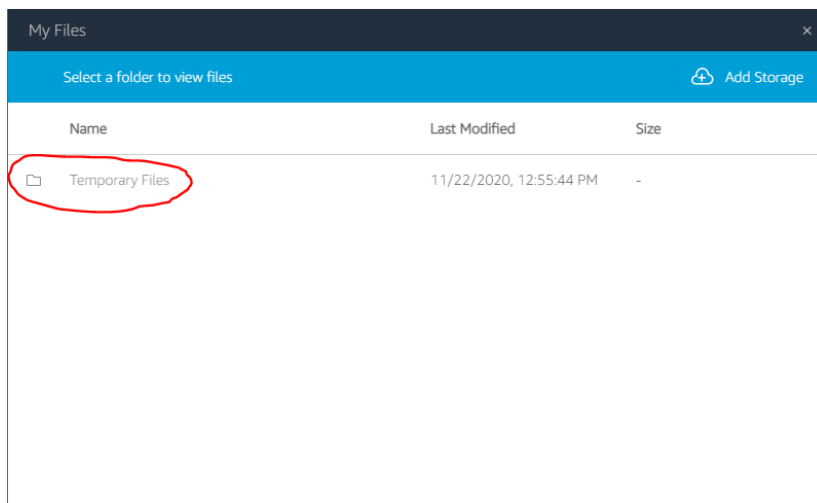
You will want to change the x and y-axis labels to match the variables we are using in this problem. You can see an example of how to do this near the end of the video on the [Numerical Results section](#) of the 2D Transient Diffusion tutorial.

Note: If you are using Apps on Demand, please use the following steps for adding a CSV file to your ANSYS results section.

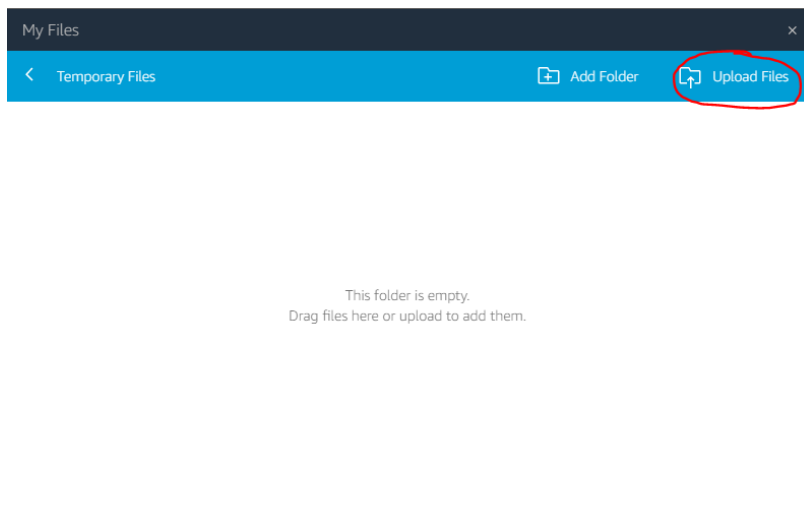
1) Click on My Files in the top left



2) Click Temporary Files



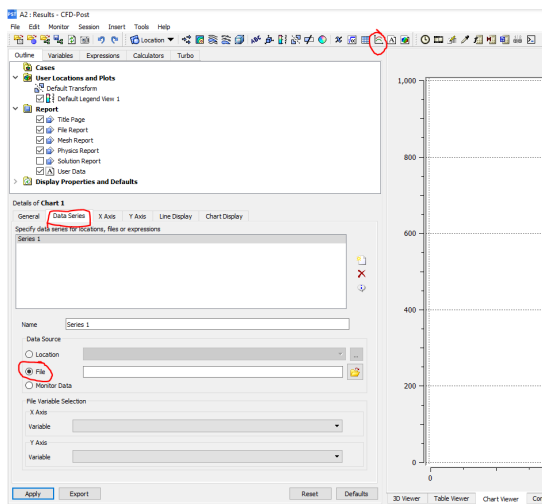
3) Click Upload Files



4) Select the CSV file from your own computer

5) Return to ANSYS CFD Post (The Results Section), create a chart

6) Under Data Series, click on file



7) Select your CSV file from the following location

PhotonUser>My Files>Temporary Files

[Go to Step 6: Verification & Validation](#)

[Go to all FLUENT Learning Modules](#)