

# 3D Steady Diffusion - Model Setup

Author(s): Keith Works, Cornell University

[Problem Specification](#)

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

[2. Geometry + Mesh](#)

[3. Model Setup](#)

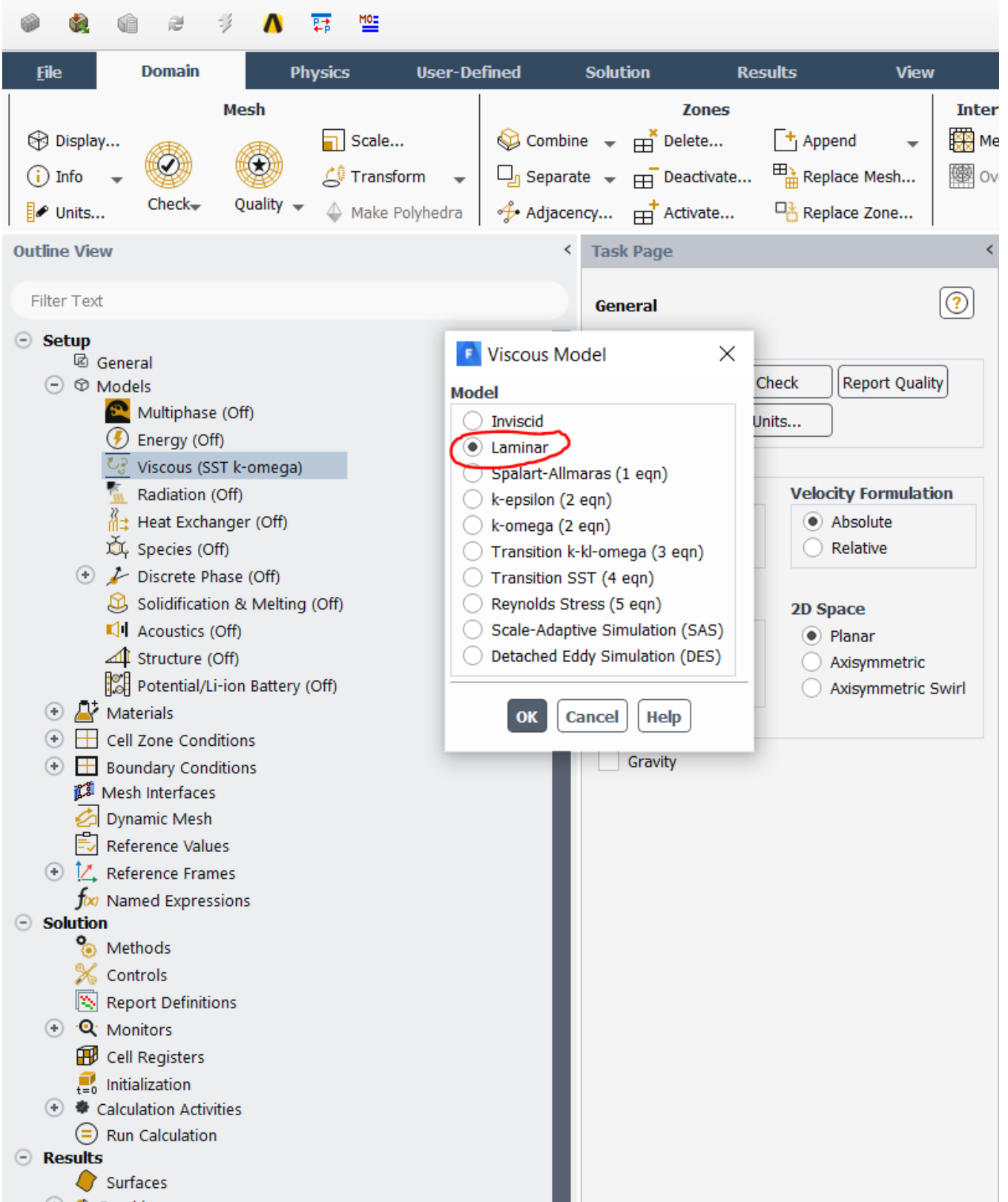
[4. Numerical Solution](#)

[5. Numerical Results](#)

[6. Verification & Validation](#)

## Model Setup

Note: Around the 25 second part of the video below, an additional step needs to be taken if you are using ANSYS software versions 2020 R1 or newer (includes Apps on Demand). In these newer software versions, the default Viscous Model in Fluent (the Model Setup step) is SST K-Omega and should instead be Laminar. To do this, simply double click on Viscous under Models and change it to Laminar.



Watch the following video for how to set up the physics in Fluent:

## **Go to Step 4: Numerical Solution**

[Go to all FLUENT Learning Modules](#)