2D Transient Diffusion - Model Setup
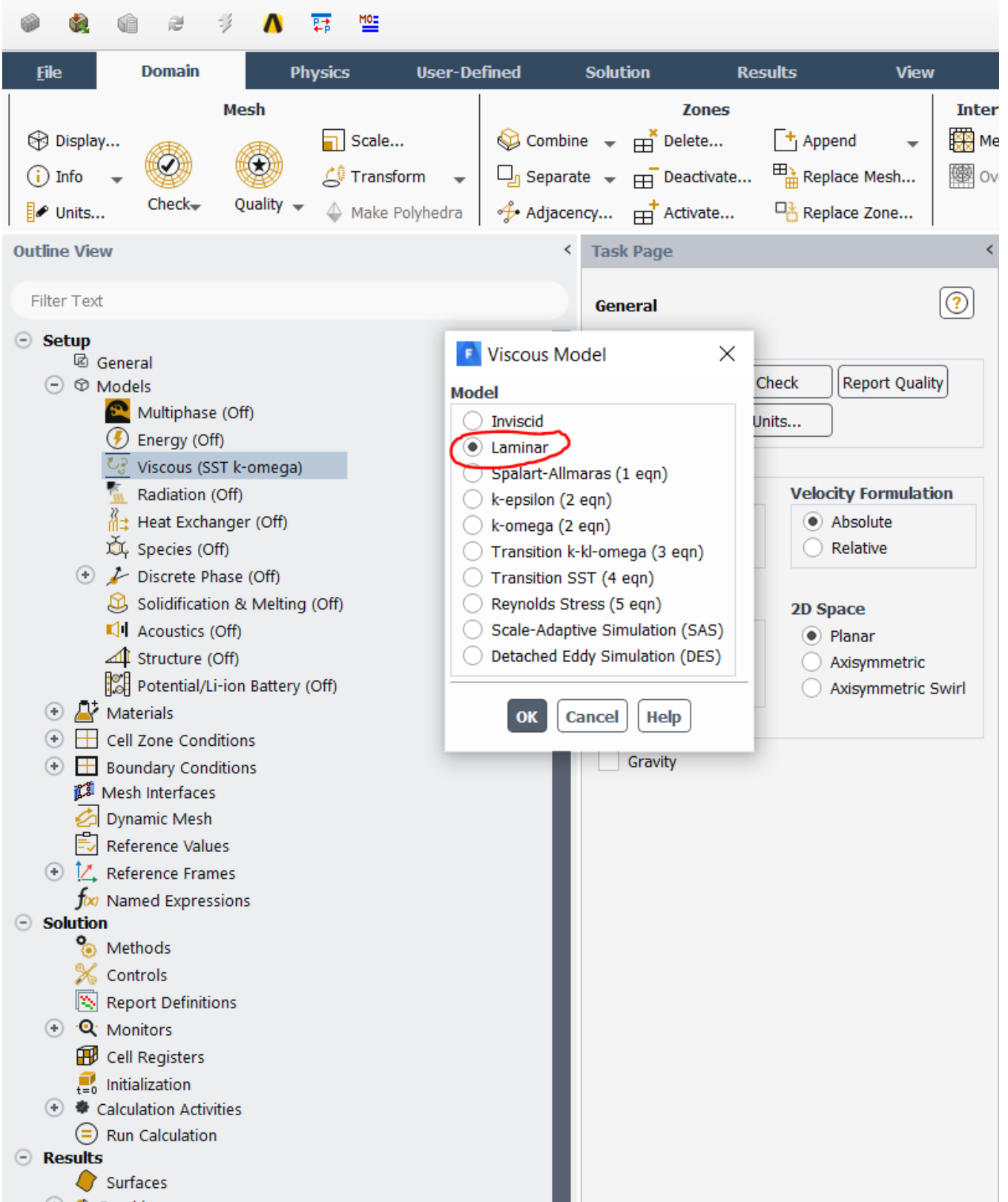
Author(s): Keith Works, Cornell University

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.

File   Domain   Physics   User-Defined   Solution   Results   View

Mesh

- Display...
- Info
- Units...

Check

Quality

- Scale...
- Transform
- Make Polyhedra

Zones

- Combine
- Separate
- Adjacency...

- Delete...
- Deactivate...
- Activate...

- Append
- Replace Mesh...
- Replace Zone...

Inter

Me

Ov

Outline View

Filter Text

Setup
- General
- **Models**
  - Multiphase (Off)
  - Energy (Off)
  - Viscous (SST k-omega)
  - Radiation (Off)
  - Heat Exchanger (Off)
  - Species (Off)
  - Discrete Phase (Off)
  - Solidification & Melting (Off)
  - Acoustics (Off)
  - Structure (Off)
  - Potential/Li-ion Battery (Off)
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
Solution
- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation
Results
- Surfaces

Task Page

General

Check   Report Quality

Units...

Viscous Model

Model
- ( ) Inviscid
- (●) Laminar
- ( ) Spalart-Allmaras (1 eqn)
- ( ) k-epsilon (2 eqn)
- ( ) k-omega (2 eqn)
- ( ) Transition k-kl-omega (3 eqn)
- ( ) Transition SST (4 eqn)
- ( ) Reynolds Stress (5 eqn)
- ( ) Scale-Adaptive Simulation (SAS)
- ( ) Detached Eddy Simulation (DES)

OK   Cancel   Help

Velocity Formulation
- (●) Absolute
- ( ) Relative

2D Space
- (●) Planar
- ( ) Axisymmetric
- ( ) Axisymmetric Swirl

Gravity

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

Go to Step 4: Numerical Solution