Tips & Tricks and Best Practices for ANSYS CFD

Madhusudan Agrawal
Bill Holmes
ANSYS Inc.
OUTLINE

• Best Practice Procedures, Tips & Tricks and Solution Strategies
  • Phase Change Models
    • Condensation, Evaporation and Boiling Models
  • Turbulence – SRS Models
  • Particulate Models
  • Porous Media Modeling
  • Non-Newtonian Flow
  • CFD Post : Tips & Tricks
CFD Seminars – Houston Office

- ANSYS Workbench
- ANSYS Meshing
- Multiphase Flow Modeling
- Turbulence Modeling
- Reacting/Combustion Modeling
- Heat Transfer and Radiation
- UDF and Customization
- Fluid Structure Interactions
- Turbomachinery

Linkedin Group: ANSYS User Group - South Texas Region
Phase Change Modeling
Evaporation-Condensation – Tips & Tricks

• Mass transfer from liquid to vapor

• Specify Latent Heat as Standard State Formation Enthalpy
  - Standard state enthalpy of vapor = latent heat (in j/kg-mol units)
  - Standard state enthalpy of liquid = 0
  - Same molecular weight for liquid and vapor
  - Reference temperature = 298.15 K

• Calculation strategy
  - Use coupled solver with low Courant numbers
  - Lower the explicit relaxation factors for pressure and momentum to 0.5
  - Ensure reverse flow volume fraction properly defined at outlet boundaries
• Tuning evaporation and condensation frequency
  – Compare the numerical results with experimental results
  – Use simple calculation to estimate evaporation
    • Evaporation expected = \( \frac{H_{\text{total}} - H_{\text{sensible}}}{\text{Latent Heat}} \)
  – Adjust evaporation/condensation frequencies (0.001 – 100)

• In Evaporation-Condensation Model, departure from saturation determines the rate of mass transfer
  – \( (T_{\text{cell}} - T_{\text{sat}}) \) is the driving force
  – For mass transfer to happen, \( T_{\text{cell}} > \) or < \( T_{\text{sat}} \)

• Increasing these frequencies
  – Predict the mixture temperature closer to saturation temperature
Boiling Models in R13

- Boiling models in FLUENT 13:
  - RPI boiling model \( \alpha_v \leq 0.3 \)
  - Non-equilibrium boiling
  - Critical Heat Flux (\( \beta \))

- Interfacial Area and Bubble Diameter
  - Algebraic formulations and UDF options
  - IAC equation compatible with boiling models (\( \beta \))

- Interfacial Transfer models
  - A range of sub-models for *drag and lift*, and turbulent dispersion
  - Liquid/vapor-interface heat and mass transfer models
  - Flow regime transitions from bubbly to droplets

Contours of vapor volume fraction in a nuclear fuel assembly
How to Use the Boiling Models?

• Choose “Boiling Model” under “Eulerian Parameters”
  • The “Energy” will be automatically turned on

• Activate the Viscous Model.
  • Boiling models only apply for turbulent flows
  • All multiphase turbulence models are compatible
  • Turn on the “Turbulent Drift Force”

• Access to “Phase Interaction” panel
  • Define Drag, Lift, Heat, and Surface Tension
  • “Number of Mass Transfer Mechanisms” as 1
  • Select “boiling” from “liquid” to “vapor”
  • Specify the “Saturation Temperature” and heat transfer coefficients
Boiling Models – Tips & Tricks

• Decide which Boiling model to choose
  – If $T_{\text{bulk}}$ below $T_{\text{sat}}$ ➔ Subcooled flow
    • Use RPI wall boiling model
  – If $T_{\text{bulk}}$ close to (within 3K) $T_{\text{sat}}$ ➔ Saturated flow
    • Use non-equilibrium wall boiling model

• Common mistakes in Boiling Model
  – Ensure gravity is ON to see any heat transfer
  – Ensure surface tension is specified
    • Needed for nucleation and growth of bubbles
  – Ensure correct phases in mass transfer mechanism

• Solution strategies similar to evaporation-condensation modeling
  – Use lower energy URF (~ 0.6)
Turbulence Modeling
Scale Resolving Simulation (SRS) Models

SRS: Resolve at least a portion of the turbulence spectrum in some part of domain (i.e. instantaneous field information)

- URANS
- Large Eddy Simulation (LES)
- Hybrid RANS/LES
  - Detached Eddy Simulation (DES)
  - Scale Adaptive Simulation (SAS)
  - Embedded LES (ELES/ZFLES)
  - Wall Modeled LES (WMLES)

Capture unsteadiness of largest scale of turbulence

- Resolves all scales of turbulence
- Tight mesh requirement

- Resolves large scales of turbulence with LES in flow separation region
- Models near wall turbulence flow with RANS
- Suitable for flows with medium and high Re number
Hybrid RANS/LES Models – DES/SAS

• In External flows around solid obstruction, flow instability quickly produce unsteady turbulence
  ▪ A favorite flow for DES or SAS
• In wall bounded internal flows, flow instability may not be sufficient to produce unsteady turbulence
  ▪ DES model --- undefined
  ▪ SAS model --- URANS or steady state solution

RANS to cover wall boundary layer

“LES” based on $\Delta$ or $L_{vk}$

Instability produced by the prism

Cylinder and Iso-surface of second invariant of the rate-of-strain tensor
(Re ~ 3 millions)
Embedded LES (ELES) in ANSYS FLUENT or Zonal Forced LES (ZFLES) in ANSYS CFX

- One or more LES zones can be embedded into a RANS zone

- LES Zone 2 is embedded in Zone 1 and 3 with RANS or SAS model

**RANS-LES Interfaces** – prescribe velocity variation based on turbulence kinetic energy $k$ from RANS (or SAS) model, with Vortex Method or Spectral Synthesizer

ELES: Spatially decaying turbulence
Hybrid RANS/LES Models - WMLES

- Wall Modeled LES (WMLES) models the inner boundary layer with RANS and resolves central part of boundary layer with LES
- WMLES reduces the near wall mesh resolution requirement especially in directions parallel to the wall surface
  - Much coarser mesh compared to full LES
  - No dependence with Re number, suitable for medium and high Re number range

Flow Over a Wall Mounted Hump

ELES with WMLES ANSYS-FLUENT

It is important to visualize Q-criterion isosurfaces to ensure that turbulent structures are OK.
Particulate Modeling
## ANSYS CFD Models for Particulate Flows

<table>
<thead>
<tr>
<th>Model</th>
<th>Numerical approach</th>
<th>Particle fluid interaction</th>
<th>Particle-Particle interaction</th>
<th>Particle size distribution</th>
</tr>
</thead>
<tbody>
<tr>
<td>DPM</td>
<td>Eulerian – Lagrangian</td>
<td>Empirical models for sub-grid particles</td>
<td>Particles are treated as point masses</td>
<td>Easy to include PSD as in Lagrangian description</td>
</tr>
<tr>
<td>DDPM</td>
<td>Eulerian – Lagrangian</td>
<td>Empirical; sub-grid particles</td>
<td>Approximate by KTG as in granular models</td>
<td>Easy to include PSD as in Lagrangian description</td>
</tr>
<tr>
<td>DDPM - DEM</td>
<td>Eulerian – Lagrangian</td>
<td>Empirical; sub-grid particles</td>
<td>Accurate calculations based on soft sphere collisions</td>
<td>Can account for all PSD physics accurately including geometric effects</td>
</tr>
<tr>
<td>Euler Granular model</td>
<td>Eulerian – Eulerian</td>
<td>Empirical; sub-grid particles</td>
<td>Approximate by KTG as in granular models</td>
<td>Different phases to account for a PSD; PB models for size change</td>
</tr>
<tr>
<td>Macroscopic Particle Model</td>
<td>Eulerian – Lagrangian</td>
<td>Accurate calculations based on local flow, pressure and shear stress distributions</td>
<td>Accurate calculations based on hard sphere collisions</td>
<td>Easy to include PSD as in Lagrangian description</td>
</tr>
</tbody>
</table>
DPM – Tips & Tricks

• Ensure volume fraction for DPM < 12%
  • Mass loading can be large (+100%)
  • Particles enter and leave computational domain

• Turbulent dispersion
  • Stochastic tracking
    • Use high # of tries (>20) for turbulent stochastic
  • Particle cloud model

• Define Injections
  • Use Rosin-Rammler distribution for PSD
  • Provide appropriate initial velocity and flow rate

• Coupled DPM - Convergence
  • DPM source term can be under relaxed
  • Perform sufficient DPM trackings for full source
  • Steady vs transient Particle tracking
  • Resetting Interphase Exchange Terms
    • Solution Initialization -> Reset DPM Sources
Eulerian-Granular: Tips & Tricks

- Calibrate drag law based on minimum fluidization velocity
- Suggested frictional settings
  - Scheaffer viscosity
  - Johnson et al. frictional pressure
  - friction packing limit of 0.55
  - solids VOF patch of 0.58
- To model turbulent dispersion for fluid-particle flows, use either of
  - `(domainsetvar domain-id 'vof/diffusion-on? #t)`
Dense DPM – Tips & Tricks

• This is an extension from DPM to account for
  – The effect of blockage on the fluid through volume fraction
  – The effect of collisions on the motion of particles through KTG
  – Very tight momentum + energy equation coupling
  – Efficient for size distributions - No penalty
  – Applicable to dense sprays, dense slurry

• Few Tips in setting DDPM case
  • Make sure injections have the particle phase as the “Discrete Phase Domain”
  • Enable Volume Fraction Approaching Packing Limit to avoid solid accumulation
  • Numerics and Discretization
    • Node-based Gradients
    • 2nd order Flow and 1st order VOF
DDPM- DEM

- DEM = DDPM + Explicit Particle-Particle Interaction
  - Soft sphere collision algorithm with friction
  - Resolved using a spring dashpot model
  - Uses parcels (not particles) for particle-particle collisions

Key Steps for DEM Setup
- Switch ON DEM Collision in Physical Model Tab in DPM panel
- Define DEM Collision Pairs – spring or spring dashpot and friction-dshf
- Set DEM Collision pair for DPM injection and DPM BCs
- Rest setup similar to DDPM setup
Porous Media Modeling
Modeling Porous Media

- Representation through momentum sinks
  - Superficial velocity based formulation
  - Physical velocity based formulation
    - Relative velocity between phases important
- A fixed multiphase phase
  - Naturally physical velocity based formulation
- Resolved porous media
  - Geometrical representation of features
- Macro Particle Model
  - Pseudo DNS type simulation to model big particles
Porous Media Model - FAQ

Porosity and viscous/inertial resistance
- Viscous and inertial resistance for momentum sink and pressure drop calculation, directional
- Porosity used in the transient term and heat conduction term only, isotropic

User defined resistances and Porosity
- Through DEFINE_PROFILE UDF macro
- Can vary with space and time
- Can be used to specify relative permeability

Turbulence modeling in the porous zone
- Toggle Laminar zone “ON” in the porous media, turbulence quantities transported through porous

Multiphase problems
- Separate resistances values for each phase
- Same Porosity for all phases and specified for Mixture

Solid Phase Volume Faction Contours
Modeling filtration
Porous Media Model – Tips for Convergence

- Patch appropriate pressure upstream and downstream of the porous zone.
- Start with first order discretization of pressure and momentum, then move to second order discretization.
- Use low URFs of pressure and momentum.
- Use PRESTO or Body-Force-Weighted scheme for Pressure.
- If the resistance coefficients are too high, start with lower values of resistance coefficients and slowly ramp them up.

Transient response

Pressure Contours

Wireline Formation Tester

- MDM for syringe action
- Compressibility of Oil
- With and without skin (mud cake)
- Multiphase - Relative permeability
Non-Newtonian Flows
Several in-built non-Newtonian (NN) fluid models
- Power law model
  - $n > 1$ for a shear thickening fluid (dilatant)
  - $n < 1$ for shear thinning fluids (pseudo-plastic)
- Carraeu model, Cross model
- Herschel Bulkley model
  - Pseudo-plastic model
- Rheopectic and Thixotropic models need UDF
  - Time dependent viscosity models
- Turbulence modification for NN fluids
  - Lam-Bremhorst low Re turbulence model
    (Damping Function)
  - `/define/models/ viscous/turbulence-expert/turb-non-newtonian`
Non-Newtonian Fluid Modeling – Tips & Tricks

- Ensure physically **reasonable limits** for viscosity
- Viscosity depends on velocity gradients (**Good mesh & numerics**)
- **Initialize** with a non-zero velocity field (**Constant viscosity**)
- Check for non-physical flow fields and viscosity **early** in the simulation
- **SIMPLEC with high URF** for faster convergence

Flow of cuttings in an Eccentric annulus (modeled as power law NN fluid with Turbulence Correction)

Comparison of Axial and Tangential velocity

Viscosity contours
Tips and Tricks
ANSYS CFD-Post
Outline

Realistic Rendering
Curved Vectors
Water Rendering
Case Comparison
Viewer State Files in Powerpoint
Realistic rendering
Half model is solved
Apply rotational instancing
Add reflection instancing
Semi-transparent circular plane
Another plane colored by Radius, Color Map = Transparency
Add shadow texture on first plane
Add metallic shading and stickers
Add plots
Add plots
Animated streamlines
Tip: Curved vectors
Tip: Curved vectors - 2

Turn on, or create a vector with the same Geometry settings.
Tip: Realistic water rendering

Iso-surface of volume fraction

Settings:

Or “Metal”
Post-processing Multiple Cases

Plot Variable Differences using “Compare Cases”
Viewer State Files in CFD Post
Graphics viewer in PowerPoint

You can display .cvf 3D graphics files generated by CFD Post in PowerPoint as follows:

First, you must enable the Developer tab in PowerPoint:
• Click the Office Button in the top-left corner of the PowerPoint window.
• Select PowerPoint Options.
• In the PowerPoint Options dialog box, enable the “Show Developer tab in the Ribbon”.
• Click OK.

To insert a cvf file into a presentation:
• Copy the cvf file to the folder where your presentation is.
• Open the presentation.
• In the Developer tab > click “Your browser may not support display of this image. More Controls”.
• Select "cfxViewer class" and click OK.
• Draw a box in the slide where you would like the viewer window to appear.
• Right-click in the viewer window and select Properties.
• In cvfFile field, type the name of the cvf file and press Enter.
• Close the Properties dialog.
Graphics viewer in PowerPoint
Questions? Time for a Break...