Ricardo Software
WAVE 1D-3D Co-simulation

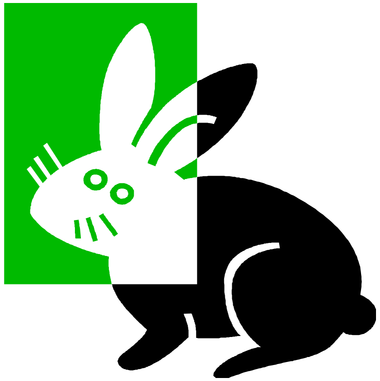
Introduction and Best Practices

Patrick Niven – 17-Jan-2013

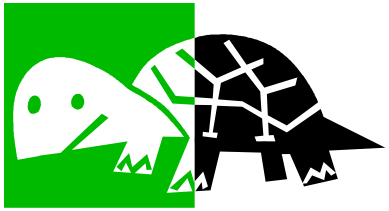
Contents

Why do it?

- WAVE solves the entire engine network quickly
  - System-wide analysis
  - Lots of basic configurations to examine
  - Eliminates minutiae from components



- 3D CFD solves individual engine components slowly
  - Why do we perform 3D CFD analysis?
  - How do we provide accurate boundary conditions?

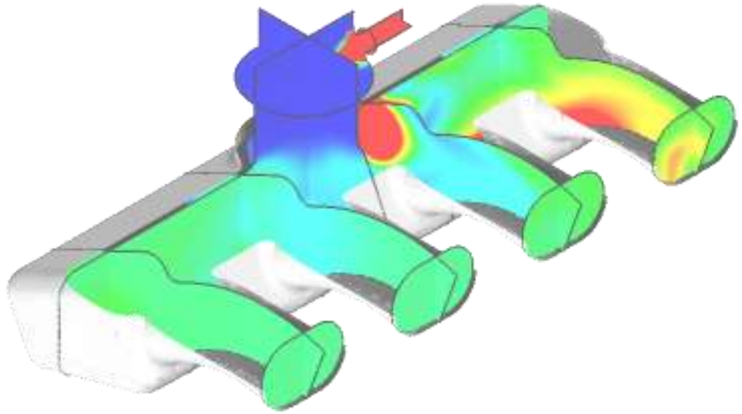


ISN'T THERE A HAPPY MEDIUM?

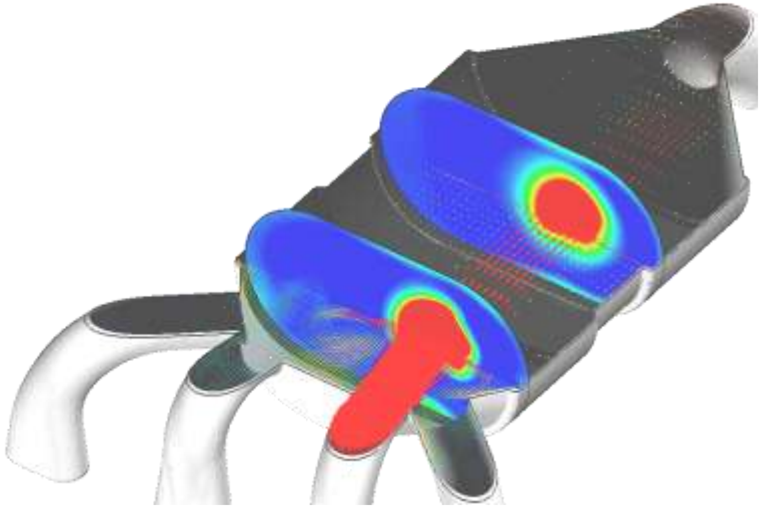
Why do it?

- In addition, there are certain physical phenomena that WAVE is not well-suited to simulate alone…

  – Secondary gas mixing
    - **EGR**
    - PCV
    - Purge



  – Complex 3D flow fields
    - Sensor locations
    - EGR tap locations
    - Catalyst brick flow distribution
    - Swirl and tumble



Why do it?

- Often the best solution is to take an existing WAVE model and replace a component with 3D-CFD geometry
  - Accurately represent the complex 3D physical phenomena
    - Fully understand 3D flow field
    - Allow detailed geometry to be developed
  - Provide realistic system boundary conditions
    - Capture feedback from 3D geometry in system predictions
  - Save time and money
    - Fewer prototype builds
    - Shorter concept cycle time

- With a well-defined process, co-simulation can often be quicker than separate WAVE and CFD simulations
  - WAVE-CFD co-simulation is the *de-facto* standard at Ricardo when both models are available

Contents

General Methodology

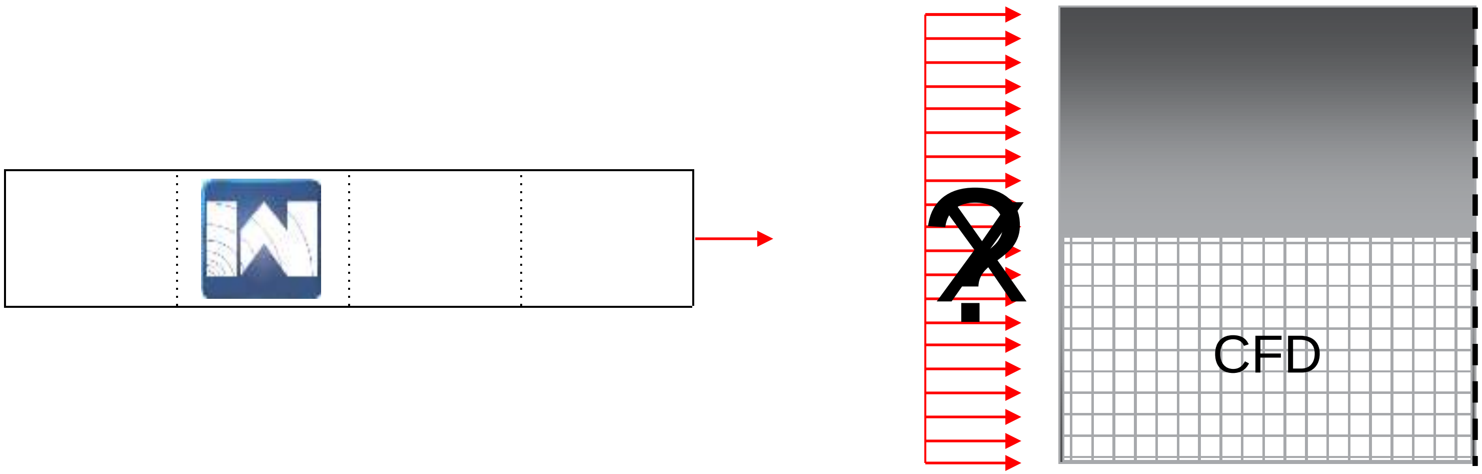
- What happens?
  1. WAVE and CFD code agree on a point in time at which to exchange data (common time step)
  2. WAVE steps forward in time, as many time steps as required to match the CFD code's time step
  3. WAVE will sub-step if necessary
  4. WAVE passes the time-averaged mass flux and last sub-volume state to CFD code
  5. CFD code steps forward in time, using WAVE boundary conditions
  6. CFD code passes the spatially-averaged boundary region states back to WAVE

- Sounds simple…IS SIMPLE!
  – Except for variations between CFD codes (not typically a single-user's problem)
    - To WAVE, it's virtually all the same regardless! Even 2-different codes!

General Methodology

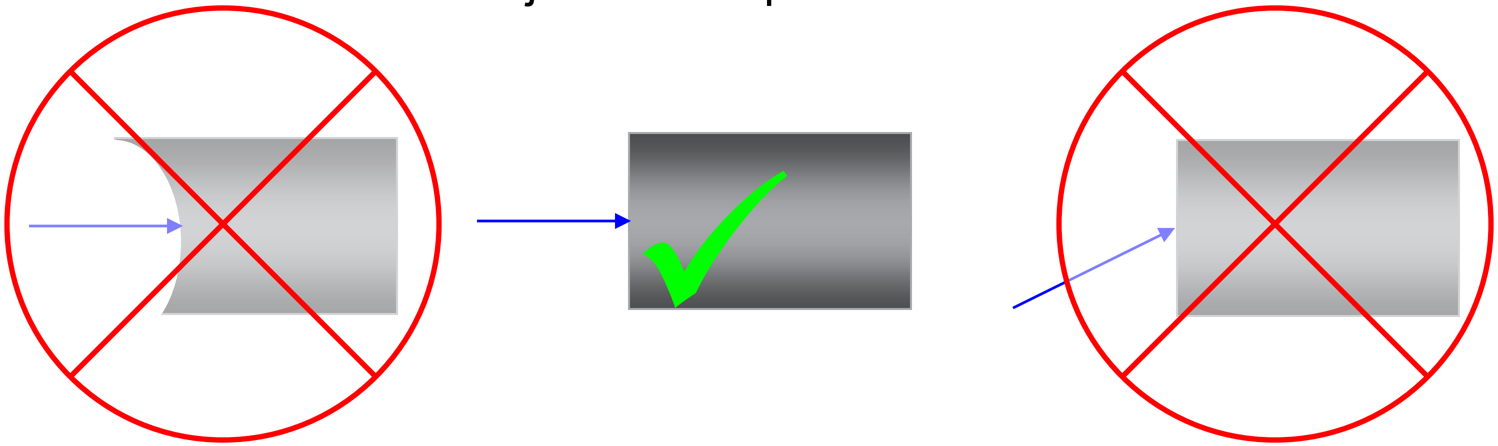
- Coupling WAVE to a CFD code requires careful thought and planning…
  - What part of the WAVE model is being substituted for 3D CFD?
  - What are the supposed flow conditions at the boundaries?

- It is assumed that the connection between the two solution domains is a flat plane normal to a region of approximately 1D flow



General Methodology

- Best place for a 1D-3D interface:
  - Plane perpendicular to the direction of flow
  - No change in cross-section area (some flexibility here…)
  - At a location that is least affected by uniform flow assumption
  - Completely unmixed or fully mixed species
    - Liquid fuel is transported from WAVE to the CFD domain as a passive scalar – it is completely non-reactive in the 3D domain (no evaporation, impingement, droplet interactions, etc.)
    - When passed back into WAVE, it magically becomes liquid fuel again
  - Remember WAVE's duct/junction requirements…



Contents

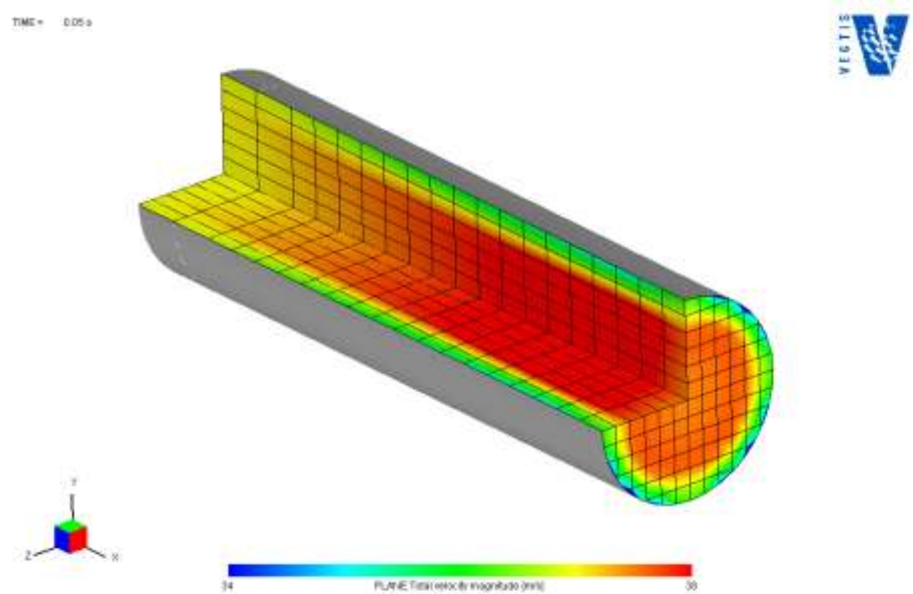
Immediate coupling

THE BASICS!

- The most basic setup is absolutely required:
  1. Place external CFD orifice junctions where coupled boundaries are desired
  2. Tell WAVE to run coupled with CFD
  3. Tell WAVE to start (or be started as a child process by) the CFD software immediately

Immediate coupling

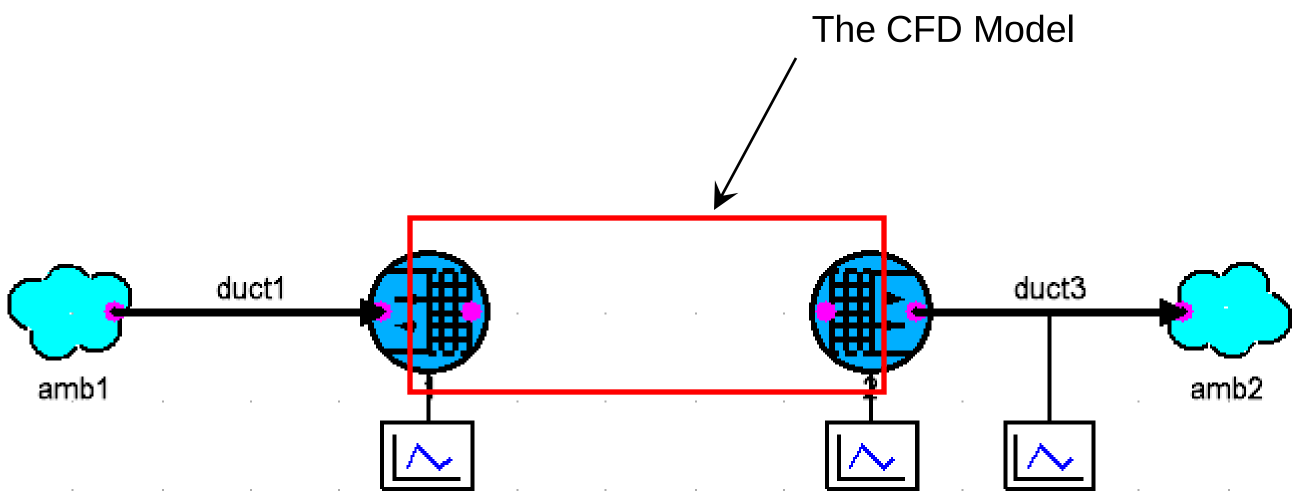
- Example #1
  - Simple pipe-flow model
  - Objective is to become familiar with the basic methodology and setup
  - Place External CFD junctions and setup CFD process
  - Examine Results
    - Compare WAVE to WAVE-CFD
    - Observe 3D flow effects in CFD results



INTERNAL UNAPPROVED DRAFT

Immediate coupling

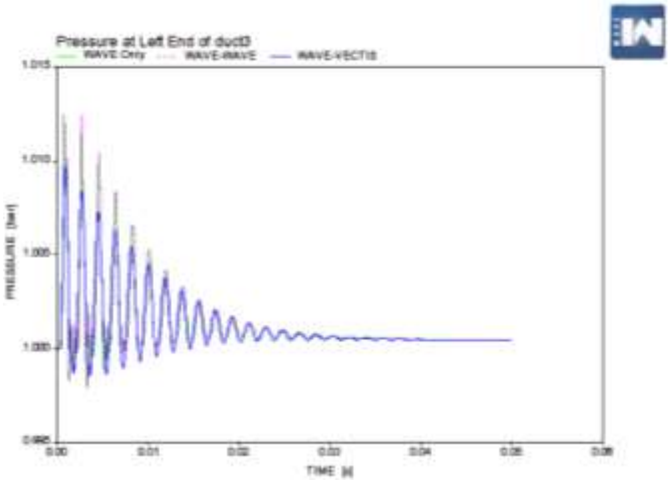
- Example #1 WAVE model

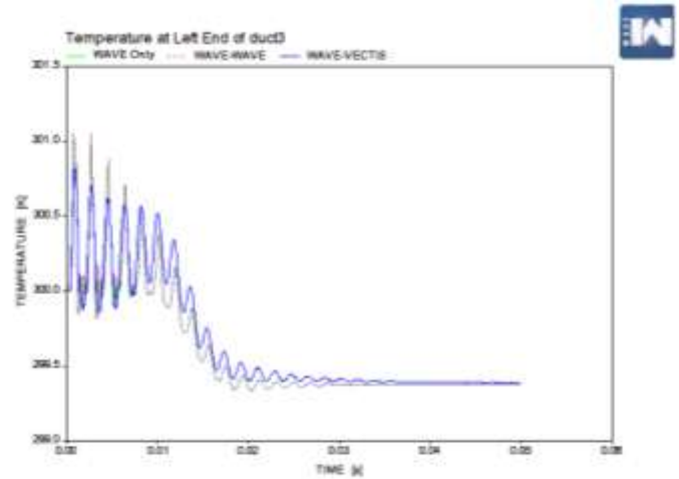


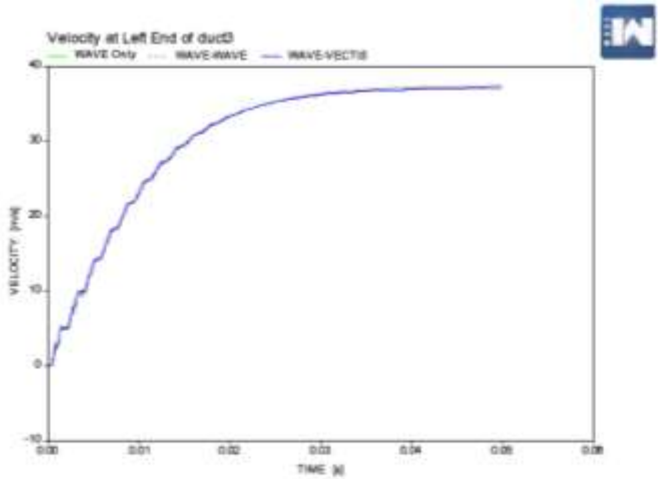
INTERNAL UNAPPROVED DRAFT

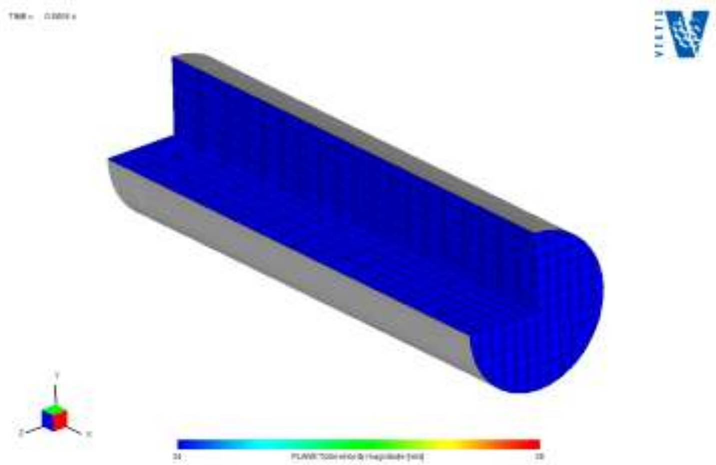
Immediate coupling

- Example #1 Results









Immediate coupling

- Ricardo Consulting Engineers has been performing coupled 1D-3D CFD analyses for 15+ years
  - In the meantime, we've learned lots of tips and tricks to improve the process!

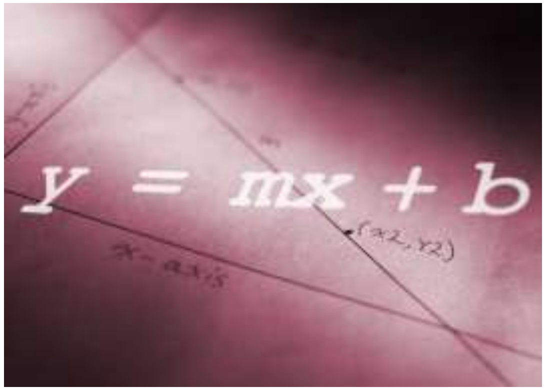
WHAT ELSE CAN WE DO TO MAKE THE PROCESS BETTER?

INTERNAL UNAPPROVED DRAFT

Two-phase coupling

THE INTERMEDIATE!

- A complex system takes time to flush out initial conditions and converge
  – WAVE can use a multi-stage coupling mechanism to initialize the WAVE model before coupling with the CFD model

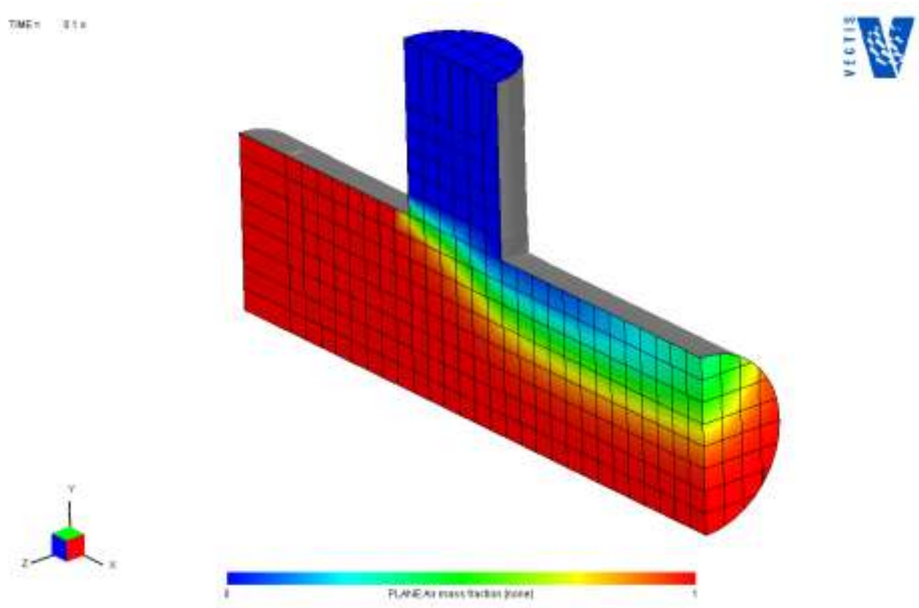


Two-phase coupling

1. Cover “The Basics”
2. Use two cases in the WAVE model (make a copy of the desired co-simulation case)
   - The first case runs a “shadow” WAVE network in place of the CFD component (doesn’t start the co-simulation) to initialize the WAVE model
     - You must have a network of representative WAVE elements between the external CFD orifice elements
   - The next case starts the co-simulation, using the final conditions from the previous case as its initial conditions
     - Make sure that the “Reinitialize model between cases” checkbox is unchecked on the “Simulation Control” panel
     - Make sure that the “Process Control” is set to “Shadow” in the first case and “Coupled” in the second case on the “External CFD Models” panel (the only change you should make between cases!)

Two-phase coupling

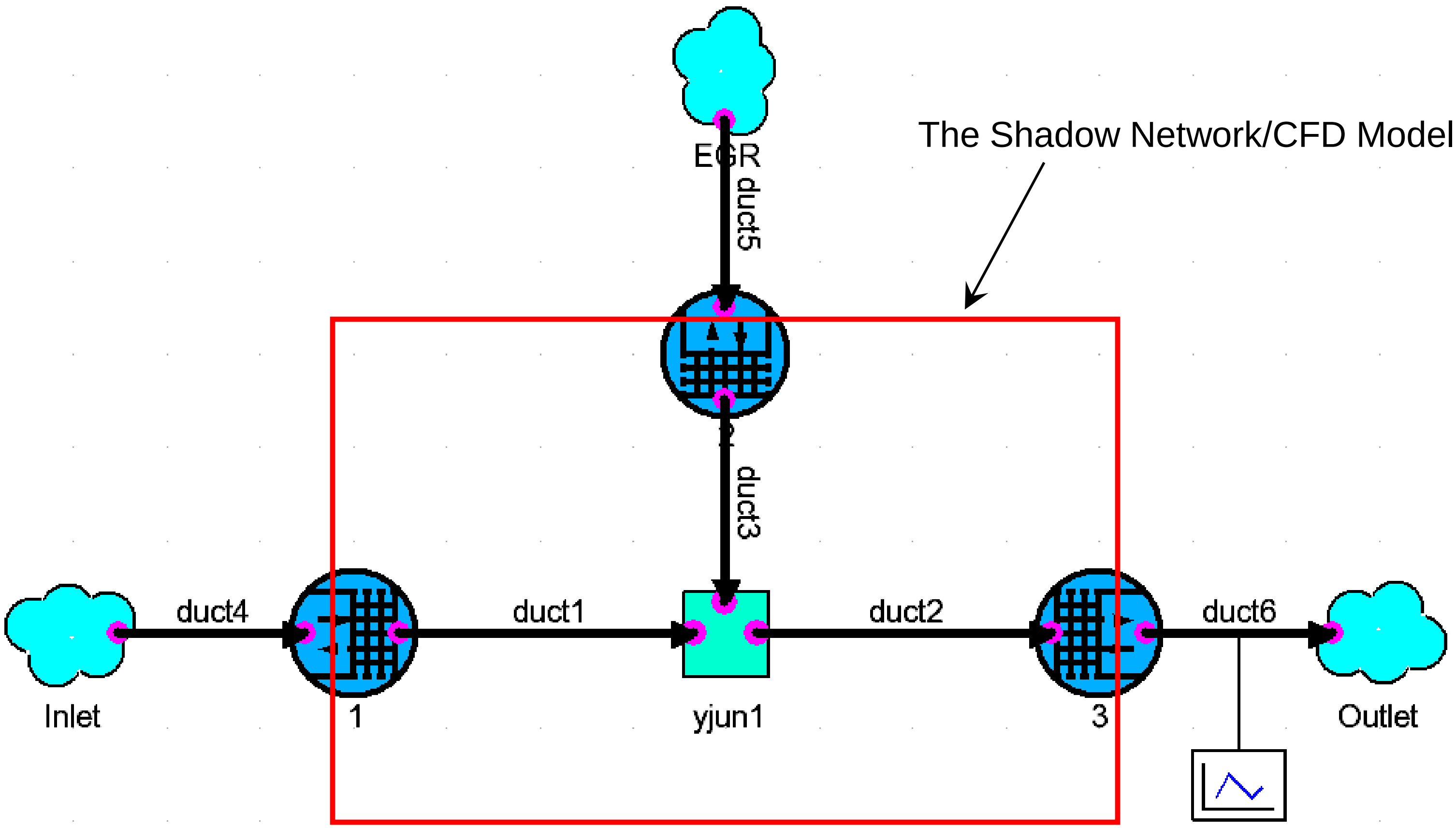
- Example #2
  - EGR Flow Model
  - Objective is to become familiar with the 2-stage methodology and setup
  - Place External CFD junctions and setup 2-stage coupling CFD analysis
  - Examine Results
    - Compare 2-stage coupling to 1-stage coupling



INTERNAL UNAPPROVED DRAFT

Two-phase coupling

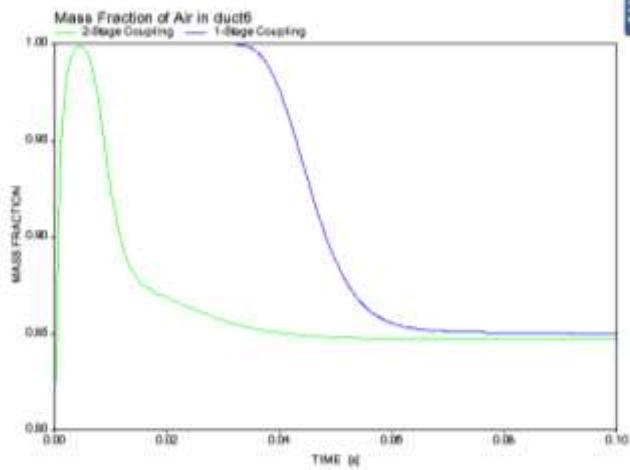
- Example #2 WAVE model

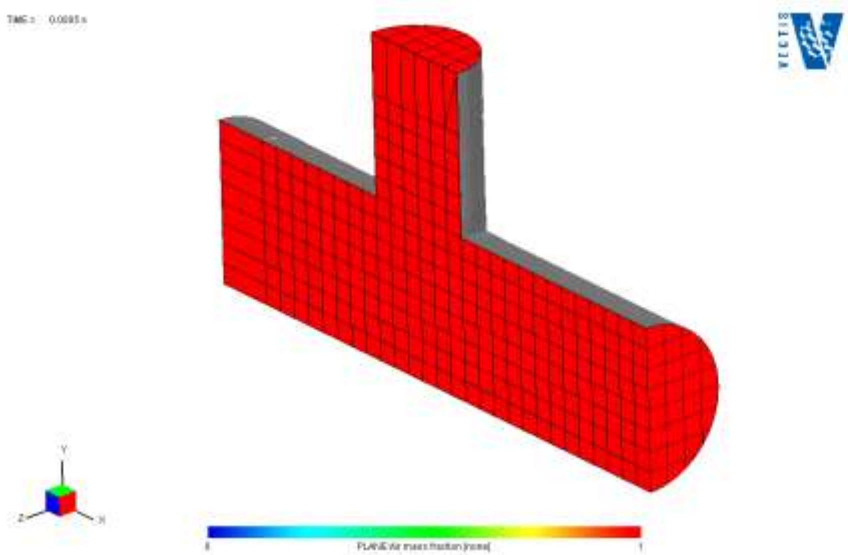


INTERNAL UNAPPROVED DRAFT

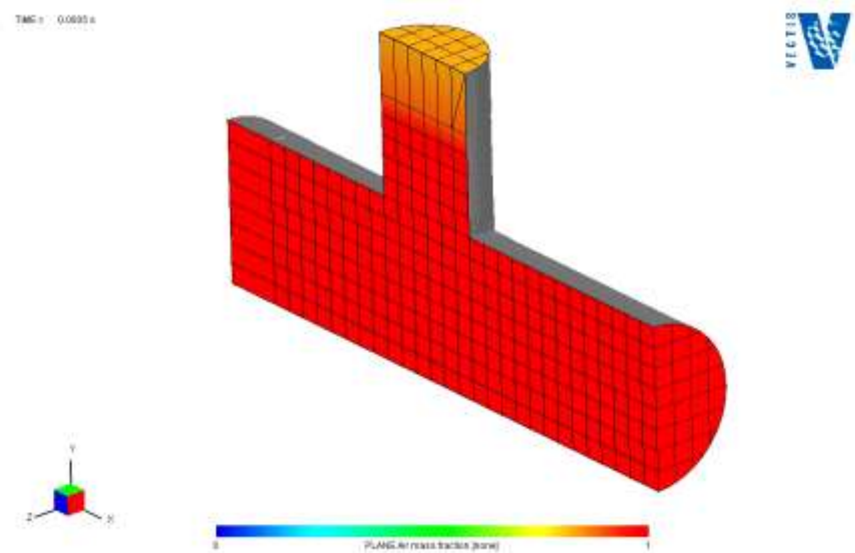
Two-phase coupling

- Example #2 Results





1-Stage Coupling



2-Stage Coupling

WAVE-CFD methodology

CAN IT GET ANY BETTER THAN THIS?

WAVE-CFD methodology

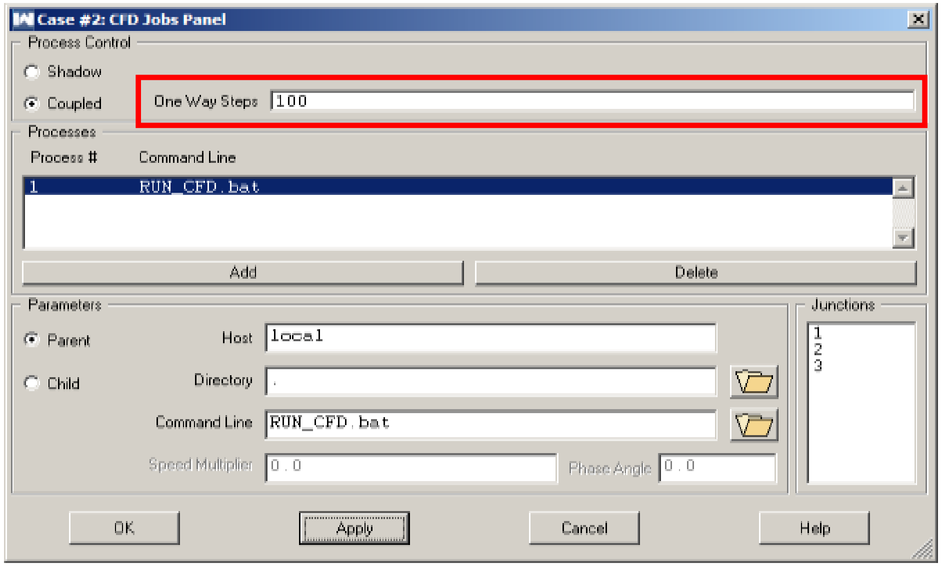
THE ADVANCED!

- Full 3-stage coupling allows the CFD model to converge as well
    1. Run WAVE model to convergence, using a representative “shadow” network
    2. Run WAVE coupled to CFD model, but only pass information one way (from WAVE to CFD)
    3. Run the fully-coupled model to convergence

WAVE-CFD methodology

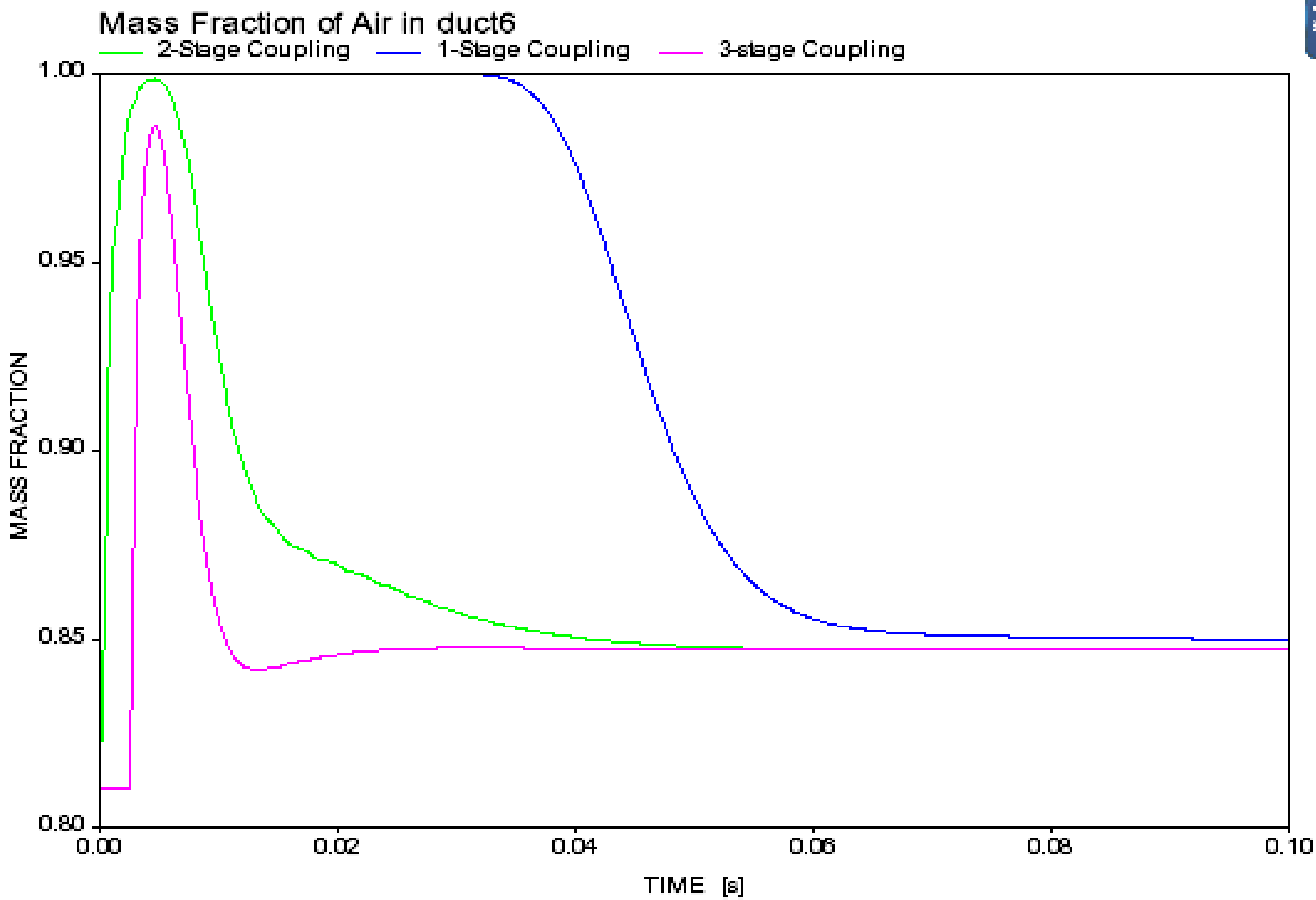
- This method allows the WAVE model to initialize in Stage 1, the CFD model to initialize in Stage 2, and the two to link and run to convergence in Stage 3

- To best illustrate this, WAVE uses 2 cases:
  - Case #1 is WAVE-only, where the WAVE model runs to convergence
    - Stage 1
  - Case #2 is coupled, with a user-specified number of one-way steps
    - Stages 2 and 3



WAVE-CFD methodology

- Exercise #2 with 3-stage coupling results!



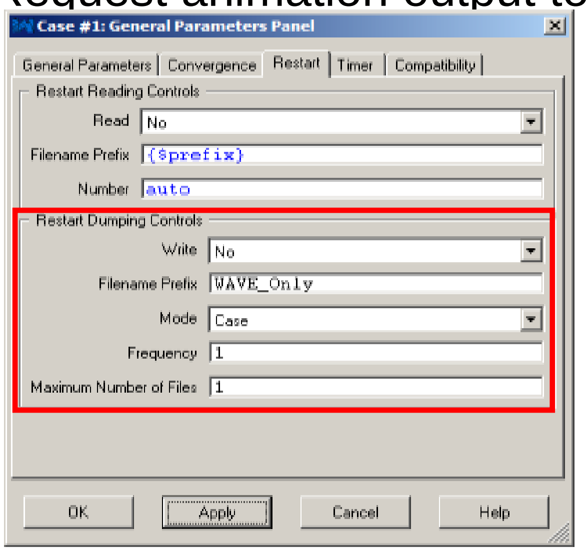
WAVE-CFD methodology

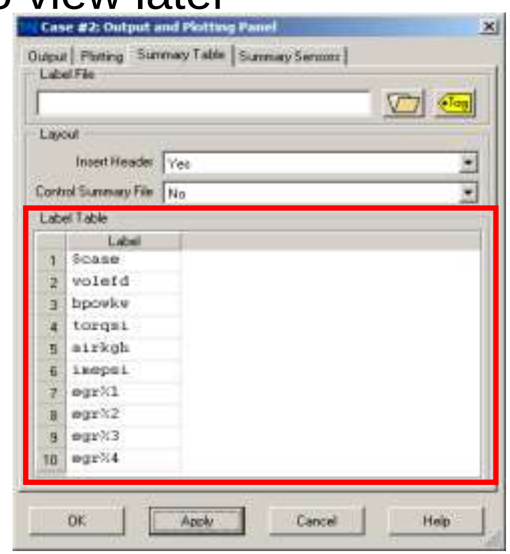
- With the introduction of WAVE3D, a new coupling mode ("Postponed") allows the exact same behaviour using just 1 case
  - Define number of coupled cycles and WAVE will run the last "n" cycles as coupled
    - Auto-convergence can be used – if achieved, WAVE skips run only the last "n" cycles as coupled

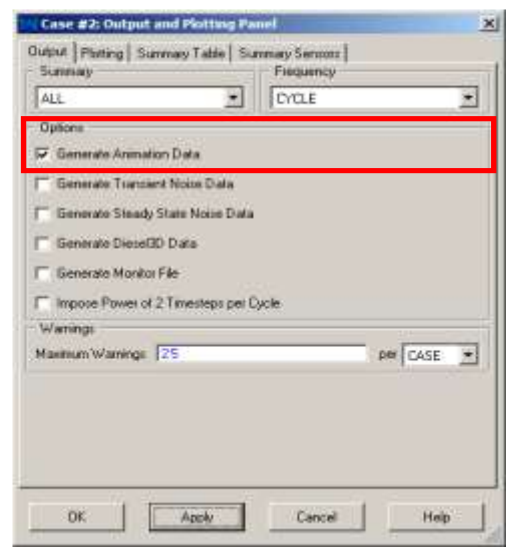
Contents

Practical issues (Real life models)

1. Run the WAVE-only case to convergence first
   – Set the model to “Write” a restart file at the end of the case (use a unique Filename Prefix)
     - This will be used to restart the model for CFD coupling
   – Set up a Summary table output
     - Pick relevant quantities which you can later use to measure the WAVE-side convergence when coupled to CFD
   – Request animation output to view later

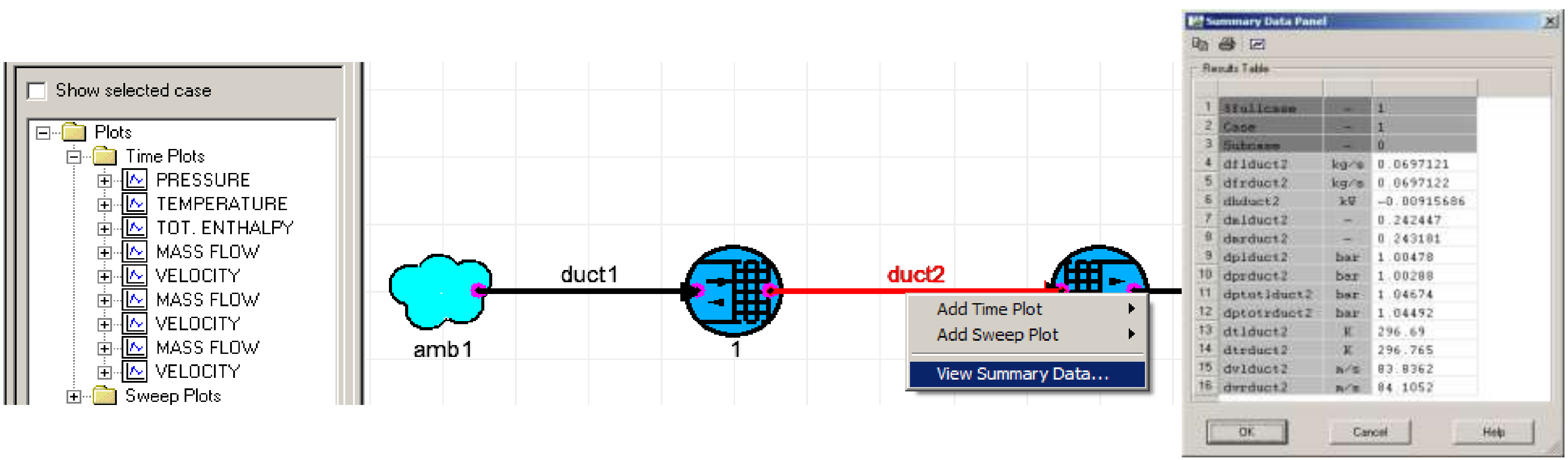






Practical issues (Real life models)

2. Examine the results in WavePost to extract information for use as initial/boundary conditions for the CFD model
   - Average pressure of ducts/junctions to be replaced by CFD domain
   - Average temperature of ducts/junctions to be replaced by CFD domain
   - Average composition of ducts/junctions to be replaced by CFD domain
   - Relevant wall temperatures of ducts/junctions to be replaced by CFD domain
   - Optionally, average velocities of ducts/junctions to be replaced by CFD domain



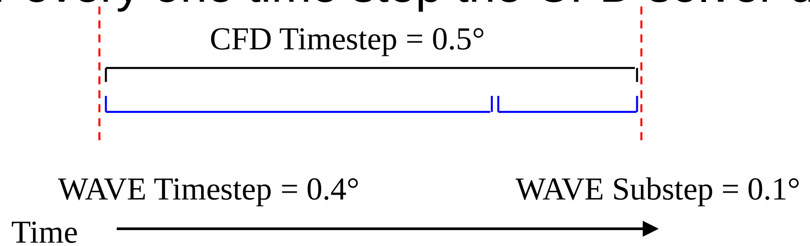
Practical issues (Real life models)

3. Use information in the .out file to calculate what CFD time step should be used
   - Search the .out file for the string “TIME STEP OUTPUT” to see how many time steps WAVE took in the last converged engine cycle
   - Divide 720 (CA° in a 4-stroke engine) by this number to determine WAVE’s average time step size in degrees
   - Calculate the CFD time step using the WAVE time step, rounded up to the nearest power-of-two degree increment (i.e. 1°, ½°, ¼°…), for example:

     WAVE time step = 0.678°, CFD Time step = 1°

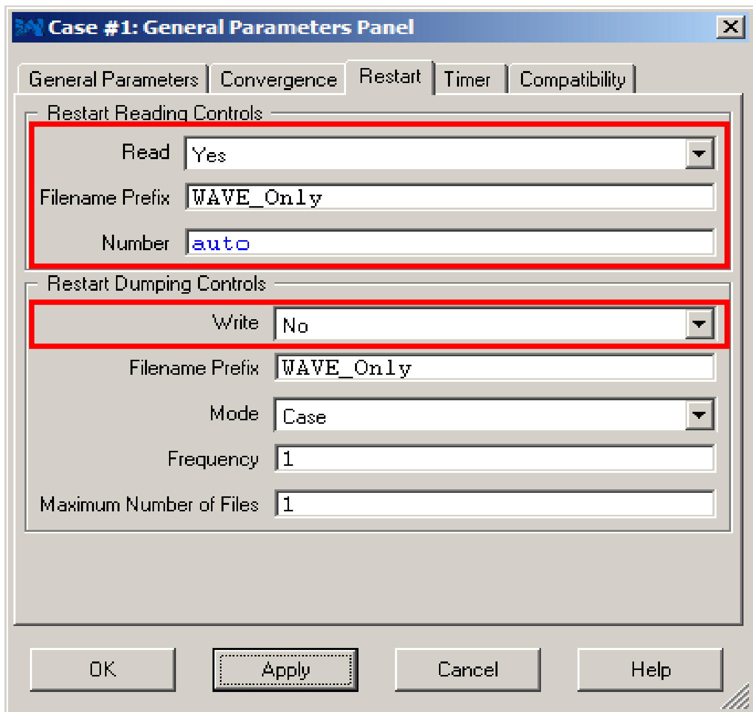
     WAVE time step = 0.424°, CFD Time step = 0.5°

   - WAVE will perform its time step(s) and sub-step if necessary to match the CFD time step so that the two simulations will be in synch. With this setting, WAVE will do two time steps for every one time step the CFD solver does.



Practical issues (Real life models)

4. Change the Restart settings in Case #1 to only "Read" the restart file and not "Write" the restart file
   – This will tell the model to restart Case #1 from where it left off – fully converged – and only run 1 more cycle, allowing it to carry the end conditions over into Case #2 as the initial conditions without having to re-run the entirety of Case #1



Practical issues (Real life models)

5. Change the Restart settings in Case #2 to only “Write” the restart file
   - The “Mode” should be changed to “Step”, indicating that the restart file will be written on the basis of user-specified number of steps in the “Frequency” field
   - The “Frequency” field should be set to match the restart file writing frequency of the CFD code (these are coupled time steps)
   - The “Maximum Number of Files” field should be set to avoid creating a massive number of WAVE restart files.

6. Set the “One-Way Steps” to run long enough to flush the volume of the 3D CFD model (for manifolds, suggest ½ engine cycle)
   - If the CFD time step is 0.5 [deg], then this number would be 720 for a 4-stroke engine

Practical issues (Restarting a WAVE model)

- There are things to keep in mind when restarting a WAVE model to run extra coupled cycles:
  - Skip Case #1 as it is no longer needed
  - Set the duration of Case #2 appropriately
    - If smaller that the completed duration, WAVE will interpret as additional time to add
    - If larger than the completed duration, WAVE will run to the specified duration (completing the difference between the two)
  - Set the number of One Way Steps to 0 (zero)
    - The simulation should be restarted in the fully-coupled state
  - Remember that WAVE always writes over any pre-existing WAVE files in the current directory
    - It may be useful to copy all of the current file into a sub-directory
  - Running an input check for a case in which a restart is set up will produce a summary of the restart file's conditions!

Contents

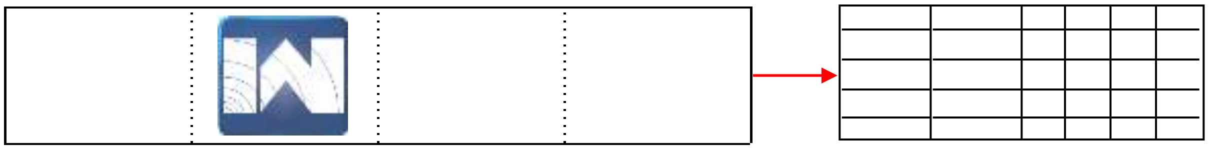
Setting up the CFD model

- CFD packages vary, but the steps are similar no matter which you use!

  1. Define link boundaries
     - Match them to boundaries as numbered in WAVE model (External CFD orifice junctions)
  2. Define wall conditions
     - Use similar settings as WAVE to make “apples-to-apples” comparison
  3. Define species properties
     - Use similar properties as WAVE (or identical when possible) to prevent step changes in density (causes unfavorable reflections)
  4. Set restart file writing frequency to match WAVE
     - Maintain consistent files (must be manually enforced!)
  5. Set duration and time step size as previously calculated from WAVE results
     - If unstable, reduce time step (too large a WAVE time step is not typically a problem)
  6. Define initial conditions to match final conditions of WAVE-only run
     - Again, prevents rapid changes in CFD model which can cause instability

Setting up the CFD model

- Some suggestions for the CFD model…
  - Increase the mesh size in the direction of flow at the linking boundaries for a few cell layers – this will help in stability as flow transitions from WAVE to the CFD domain
    - Typical WAVE dx is 30+ [mm], typical CFD dx is < 10 [mm]

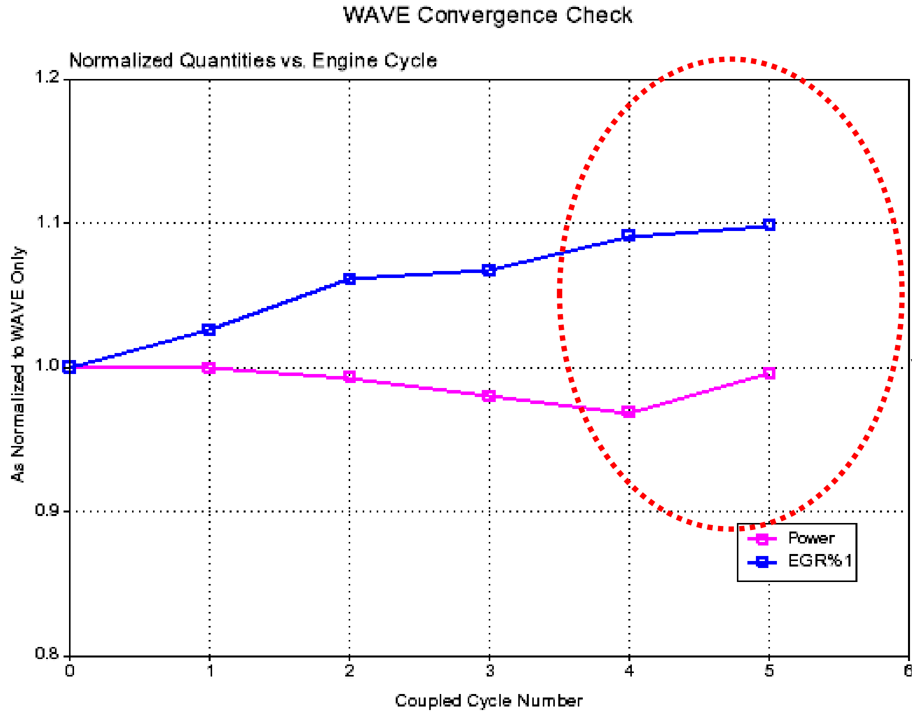


  - Consider ramping up the flow at the boundaries of the CFD model if even the one-way coupling portion of the analysis causes the CFD model to crash
    - Sometimes the initialization of the CFD model with stagnant air means that the first few one-way steps can cause too severe of a “shock” to the CFD model – ramping up flow to match the first time step’s boundary conditions means that one-way coupling will start the CFD model with very little “shock”.

Contents

Judging convergence

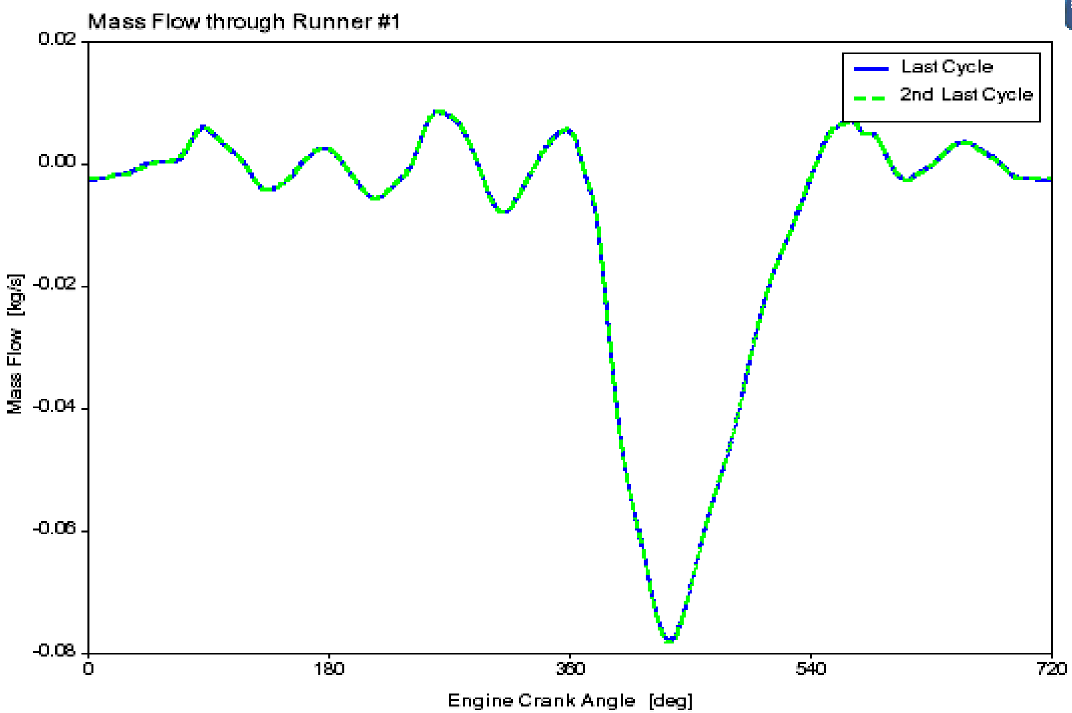
- Plotting WAVE summary values
  - Plot your relevant summary quantities vs. cycle to determine when WAVE is producing cyclically-consistent values. If your output was set to “CASE” in Case #1 and “CYCLE” in Case #2, then you can normalize against Case #1 values to see percent change in the coupled simulation vs. WAVE-only.



These lines should be “horizontal” when WAVE is converged

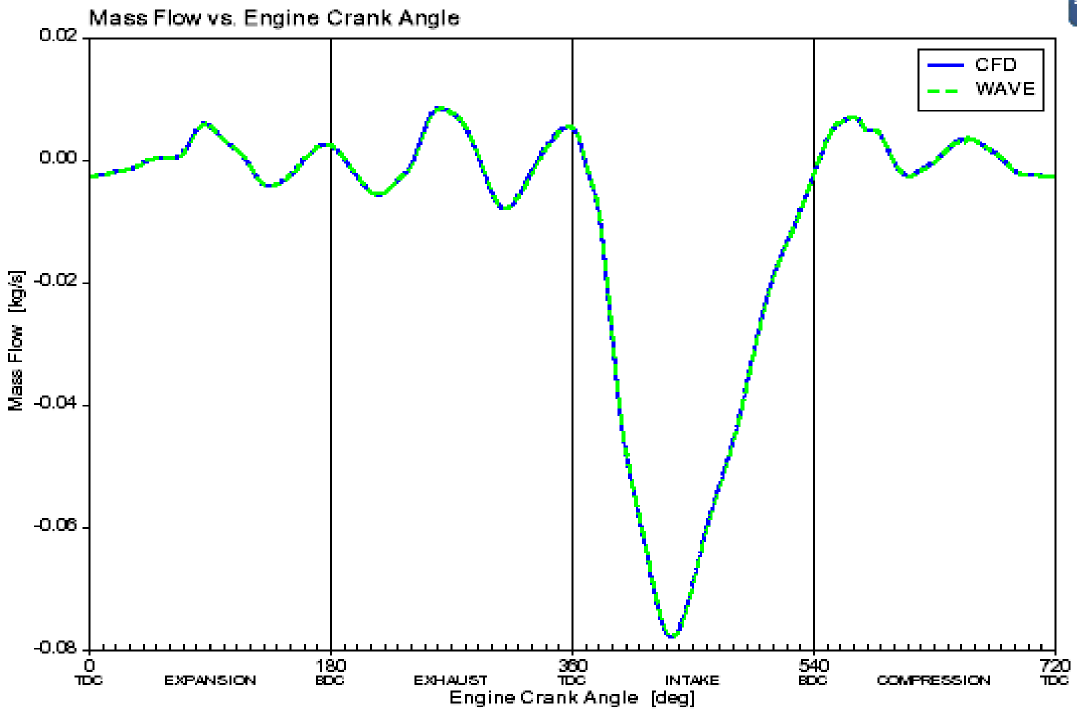
Judging convergence

- Plotting CFD boundary flow quantities
  - If your CFD model dumps boundary data on a timestep basis, you can plot relevant quantities vs. timestep/crank angle/time (crank angle/time is best in case your time-step is varying). When these conditions are cyclically-repeating, the CFD model is converged.



Judging convergence

- Using WAVE timeplots
  - If you’ve set WAVE to create animation data, relevant conditions at the External CFD orifice boundaries can be plotted vs. crank angle. When these conditions are cyclically-repeating, the CFD model is converged.
    - These should be identical to the values used by the CFD model and can be double-checked

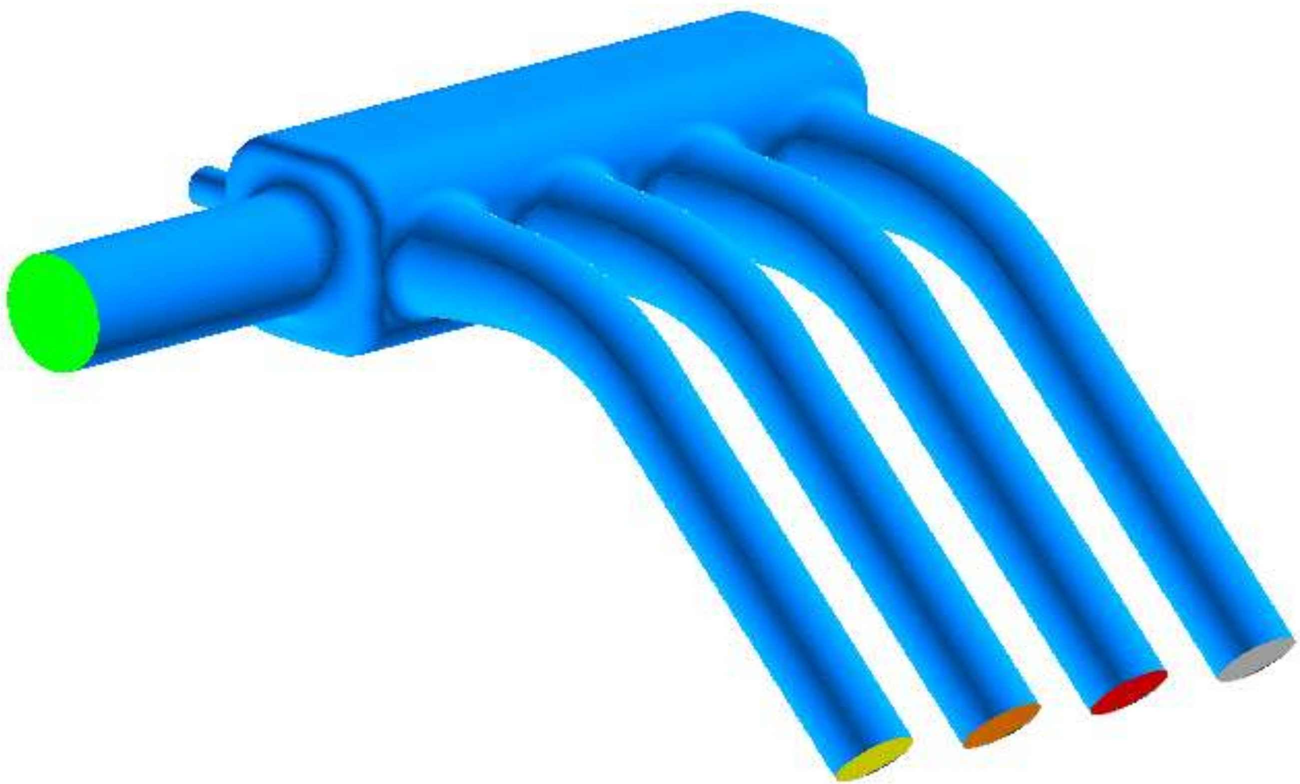


Judging convergence

- Things to be aware of:
  - Mass flow across coupled boundaries should be checked with temperature (really, this is similar to enthalpy)
    - Just because mass flow is cyclically repeating, doesn't mean that the model is converged…species concentrations may still be changing
  - WAVE model typically converges first, but not always
  - Manual inspection is best, thus turning off auto-convergence check in WAVE is recommended
  - Check all coupled boundaries…boundaries closer to engine will typically converge first (e.g. runners before inlet in intake manifold, exhaust ports before collector in exhaust manifold), but not always

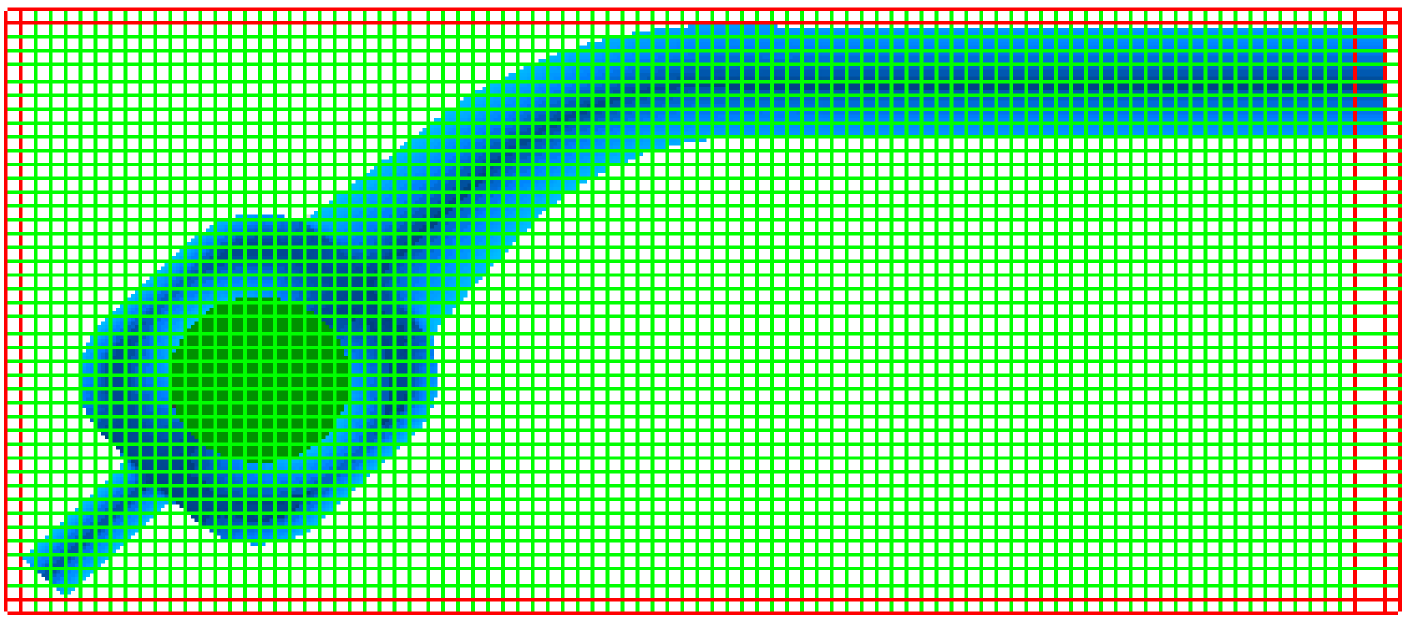
Intake manifold EGR example

- Example from Exercise #3
  - EGR intake manifold from an in-line 1.6L 4-cylinder engine
  - Objective is to analyze cylinder-to-cylinder EGR distribution



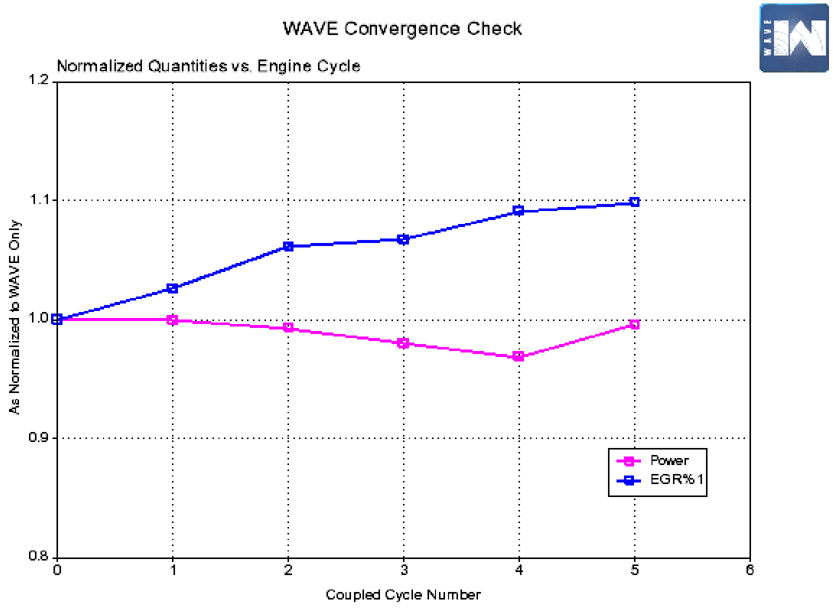
Intake manifold EGR example

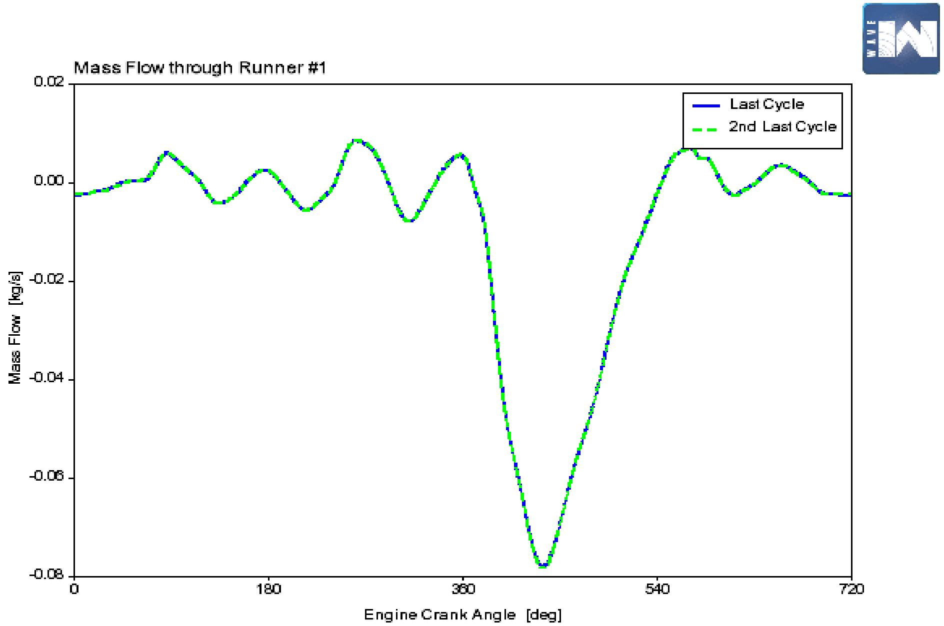
- CFD model has coupled boundaries 1-6
  - Coupled at intake ports 1-4 as link number 1-4, respectively
  - Intake boundary coupled as link number 5
  - EGR boundary coupled as link number 6
- Mesh is aligned with intake ports (maximum velocities expected here)
  - 5 [mm] global mesh size, 10 [mm] for one cell layer at this interface



Intake manifold EGR example

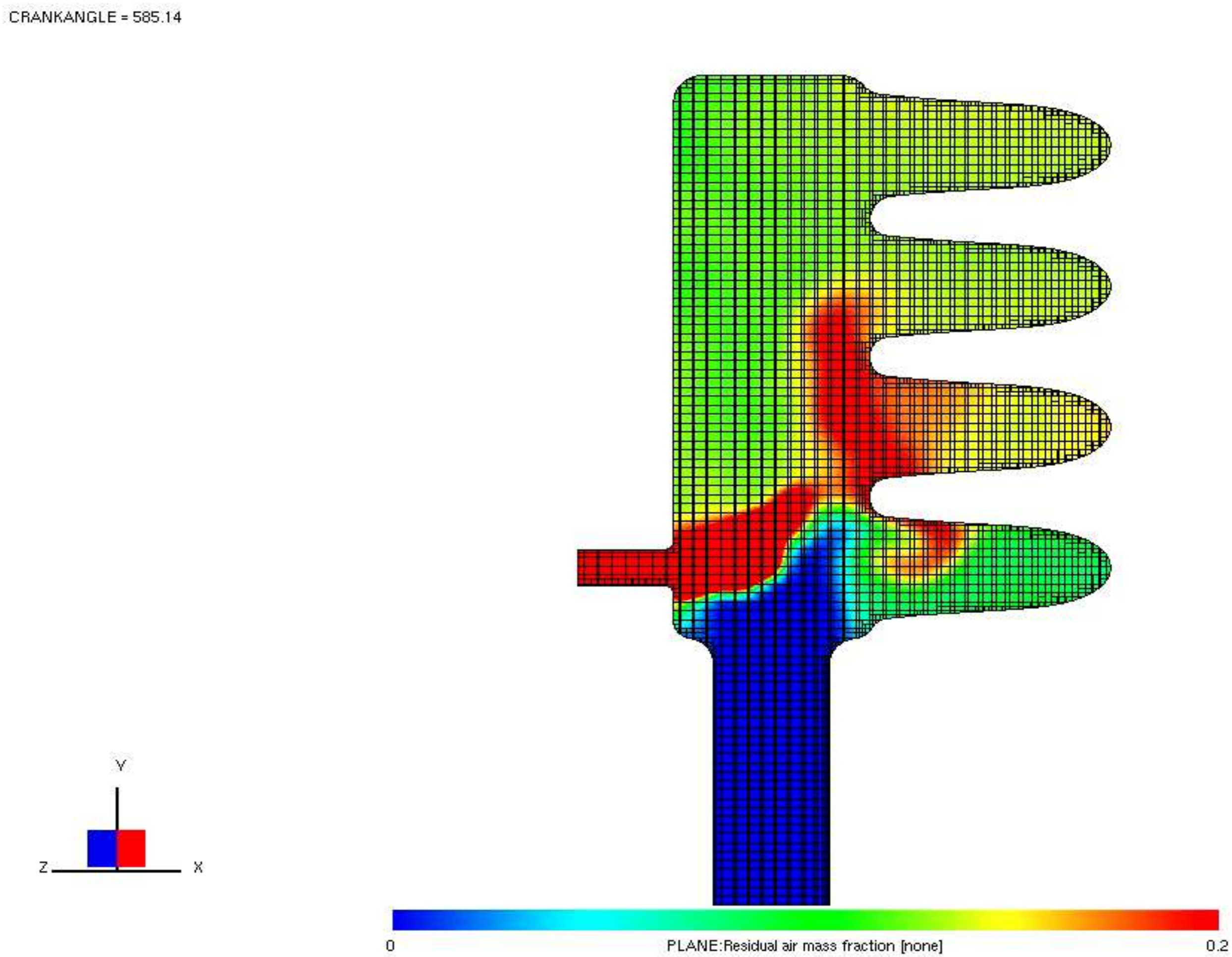
- Set up 3-stage coupling analysis with 5 coupled cycles requested in Case #2
  - After 5 cycles convergence was checked:
    - Mass flows vs. CA look converged, but EGR distribution isn't quite
  - Restarted to run 1 more coupled cycle with post-processing dumping





Intake manifold EGR example

- Visualize to understand why the distribution is as plotted…



Contents

WAVE3D Advantages

- Given that Ricardo Software markets both 1D and 3D CFD tools, as you might imagine, there are advantages to using these tools together (vs. any other 3D CFD tools)
  - VECTIS can model liquid fuel passed from WAVE as fuel droplets
  - VECTIS can use WAVE’s .fue files to easily define species
  - VECTIS will align its starting time to match WAVE’s end-of-cycle angle automatically
  - WAVE can co-simulate with both VECTIS and VECTIS-MAX solvers

- Even tighter integration is available through the WAVE3D module
  - This module is a collection of enhancements directly in WaveBuild and WaveMesher which allow you to set up a WAVE-VECTIS co-simulation without needing to leave the WAVE GUI environment
    - Easy to use
    - Inexpensive 3D CFD solution for co-simulation

WAVE3D Advantages
Setting up the 3D CFD Model

- CFD packages vary, but the steps are similar no matter which you use!
  1. ~~Define link boundaries~~
     - ~~Match them to boundaries as numbered in WAVE model (External CFD orifice junctions)~~
  2. ~~Define wall conditions~~
     - ~~Use similar settings as WAVE to make “apples-to-apples” comparison~~
  3. ~~Define species properties~~
     - ~~Use similar properties as WAVE (or identical when possible) to prevent step changes in density (causes unfavorable reflections)~~
  4. ~~Set restart file writing frequency to match WAVE~~
     - ~~Maintain consistent files (must be manually enforced!)~~
  5. Set duration and time step size as previously calculated from WAVE results
     - If unstable, reduce time step (too large a WAVE time step is not typically a problem)
  6. Define initial conditions to match final conditions of WAVE-only run
     - Again, prevents rapid changes in CFD model which can cause instability

WAVE3D Advantages
Setting up the 3D CFD Model

- Additionally:
  - Single case (Coupled mode=postponed) is used, allowing auto-convergence to be used
  - External CFD job is automatically set up per case, allowing multiple cases to be run back-to-back

  - Enhancements in WaveMesher mean:
    - Shadow network and 3D CFD mesh are both created within single GUI
    - Reference point and Monitoring points can be selected interactively on the 3D CFD geometry
    - No need to define mesh bounding box
    - Cartesian mesh can be quickly aligned to the desired I/O boundary

WAVE3D Advantages
Setting up the 3D CFD Model