Summary of dual CESE (dCESE) verification cases and sample examples

Part-I: verification cases

In this part, there are 12 verification cases, and the dual CESE (or dCESE) solver results will be compared with analytical, experimental, or results from reference papers to show the high accuracy and high shock resolution of the dual CESE method.

dCESE-CFD-2D-001v: shock-reflection

in this test, an oblique shock wave created by a shock tube is reflected on a solid wall. This is a standard CFD test case since it has an exact solution. Here we choose it as our first test case to show the high shock resolution of the CESE method. Of course, we also compare our dCESE results with the exact solutions.

dCESE-CFD-2D-002v: shock-diffraction

In this test, the diffraction of shock waves around a planar wall corner is simulated, and the dCESE solver results are compared with experimental ones at three time levels. These results demonstrate the high shock resolution of the dCESE solver.

dCESE-CFD-2D-003v: shock-over-ramp

In this test case, a supersonic flow over a 15° ramp is modeled. After the flow reaches steady state, the flow Mach number, temperature, and density at a specified point near the flow exit will be compared with the target values to show the accuracy of the dCESE solver.

dCESE-CFD-2D-004v: flow-over-corner

In this test, a centered expansion of inviscid supersonic flow around a corner is modeled by the dual CESE solver, and the Mach number Downstream of the Corner (after Expansion) is compared with the target (analytical) one.

dCESE-CFD-2D-005v: forward-facing-step

This test involves a transient, compressible, and inviscid flow, and the resulting shock reflecting and triple point. The dual CESE results compare well with a reference paper's results to show the dual CESE method's high shock resolution capability without any special treatment.

dCESE-CFD-2D-006v: shock-bnd-layer

In this test, a strong shock impinging on a flat plat is simulated by solving the viscous N-S (Navier Stokes) equations. In order to resolve the boundary layer, a clustered mesh near the solid wall must be employed. Boundary layer separation, two reflective shock waves and one compressive wave can be seen at the shock impinging point. Pressure distributions and skin friction along the flat plate are compared with the experiment data.

dCESE-CFD-2D-007v: CD- nozzle-2D

In this test case, a steady state flow in a Convergent-divergent (CD) nozzle or so-called de Laval nozzle is calculated by our dual CESE solver, and the results are compared with the analytical ones, including the shock wave location.

dCESE-CFD-2D-008v: flow-RAE2822-airfoil

In this test, a compressible (transonic), turbulent flow over the supercritical airfoil RAE 2822 is simulated, and the pressure coefficient distribution on the airfoil surface, lift and drag coefficients are compared with the experiment data to show our dCESE solver capabilities for turbulent flow simulation, even if there is no turbulence model used.

dCESE-CFD-3D-001v: CD-nozzle-3D

The flow in a 3D convergent-divergent (CD) nozzle (or de Laval nozzle) is simulated and the results are compared with the analytical ones, including the shock wave location.

dCESE-2PHASE-2D-001v: shock-bubble-interaction

This example is to test one of our multiphase models (i.e., two-phase model) newly implemented in the dual CESE solver. In this test case, a left moving shock wave propagates first through the stiffened gas before it hits a stationary bubble of a van der Waals fluid. After the shock wave hits the bubble, circular pressure waves are created, while a pair of vortices appears on the tail of the bubble. We can see the whole interaction process between shock wave and bubble, and we also compared our results with a reference paper's results at three separate times.

dCESE-HYBRID-2D-001v: rate-stick-ideal-gas

This example is to test one of our multiphase models (i.e., hybrid model) newly implemented in the dual CESE solver. In this test case, a slab of condensed explosives in a stick form, confined by an inert material, is ignited by a booster and a detonation wave propagates through the slab of the reactive material. The results are compared with those of some published papers.

Note: in this test case, an ideal gas EOS is used for all three materials (i.e., explosives (reactant & product) and inert confiner), and a simple pressure-based reaction rate law is assumed.

dCESE-HYBRID-2D-002v: rate-stick-JWL

In this test, most of the setup is similar to the above problem (i.e., dCESE-HYBRID2D-001). The only differences are a more realistic JWL EOS is used for all three materials; and the Ignition and Growth (I&G) reaction rate law is assumed. The dual CESE solver results will also be compared with those from a reference paper.

****

Part-II: More dCESE Examples

In this part, we will give more dCESE examples, especially fluid/structure interaction (FSI) examples. These demonstrate the wide range of applications of our dCESE solver.

dCESE-CFD-2D-001: Flow-arc-airfoil

This test example is designed to show users how to add a new flow direction option in the non-reflection boundary condition setup card when this boundary surface is not normal to the flow direction. If the direction is given correctly, the boundary treatment perturbations to the main flow will be significantly reduced in some cases.

dCESE-CFD-2D-002: Flow-in-tube-w-holes

In this test, we will show users that different outlet boundary setup will have different effects when a non-reflective boundary condition is used, since it is especially important to correctly set up the boundary condition especially when this boundary surface is not far enough from the main flows.

dCESE-CFD-2D-003: Shock-wedge-automesh

This example demonstrates how to use the automesher in the dCESE solver. Please note that our dual CESE solver (and the older LS-Dyna CESE solver too) will honor meshes made up of different element types, such as triangle, quadrilateral, or a mixture of triangles and quadrilaterals for 2D. In 3D, the mesh can be made of a mixture of tetrahedra, hexahedra, and wedge/prism elements. This mesh can be provided by the user, or it can be generated by the automesher as part of the setup stage of a simulation with LS-Dyna. When using the automesher, the user provides the enclosing watertight surface meshes. Currently, the automesher can only generate triangle (2D) and tetrahedral (3D) meshes.

dCESE-CFD-3D-001: Flow-in-rotating-ring

This example is specially designed to show users how to add a rotating option in our solid wall boundary condition card. Please note that it only works for viscous flows. Also, this example setup computes a steady-state flow. Given that our dual CESE (and regular CESE) solver uses an explicit time-integration scheme, it is designed for transient compressible flows. That is, it is not very suitable for such steady-state and low speed flows.

dCESE-CFD-3D-002: Supersonic-flow-rocket

The dual CESE method is specially designed for transient compressible flows, especially for high-speed flows with complicated shock waves. This example demonstrates a supersonic flow around a rocket; we can see the complex shock pattern around the rocket body.

dCESE-2phase-3D-001: Shock-bubble Interaction-3D

In this example, the previous 2D 'shock-bubble Interaction' test case in part-I (dCESE-2PHASE-2D-001v) is extended to a more realistic 3D case. The 3D effects can be seen if you compare the 3D results to the 2D results.

dCESE-hybrid-3D-001: Rate-stick-JWL-3D

In this test case, the previous 'detonation propagation in a JWL rate stick with confinement' in part-I (dCESE-HYBRID-2D-002v) is extended to the 3D case to test our 3D hybrid multiphase model solver.

dCESE Fluid/Structure Interaction (FSI) examples

dCESE-FSI-ibm2D-001: Pole-deformation-2D

In this test, a structural beam (pole) stands in the middle of a flow field with its bottom end node fixed. When a strong wind flows from left to right, it pushes the pole and forces it to deform. This example shows our dual CESE FSI-ibm solver that is based on the immersed boundary method (ibm) capability.

dCESE-FSI-ibm2D-002: Shock-wedge-interaction

In this 2D FSI-ibm test case, a shock wave (initially located at the left boundary) is moving from left to right while a rigid body structural wedge is moving in the opposite direction. From the simulation results, we can see a complicated flow pattern because of the interactions between the off-body shock produced by the moving wedge, the upcoming shock, and the wedge.

dCESE-FSI-ibm2D-003: Air-gun

In this test, a bullet is initially located inside a chamber with air at rest, and a high-pressure air chamber is located behind it. Both chambers are separated by a diaphragm. Once the separating diaphragm is removed, a high-pressure air jet flows into the chamber containing the bullet and pushes the bullet out rapidly.

dCESE-FSI-ibm2D-004: Flying-bullet

In this test, a flying bullet in a shot tube is simulated after it is triggered. The flow field resulting from this high-speed flying bullet is simulated in this example.

dCESE-FSI-ibm3D-001: Folded-bag-deployment

In this 3D FSI example, a simplified airbag is folded initially. Then, when a high-pressure jet from the entry tube is injected into the bag, the bag will rapidly deploy (inflate). We can see the entire process of this deployment and we can also see the flow field changes inside and outside the airbag with the bag deploying.

dCESE-FSI-ibm3D-002: Solid-shell-deformation

In this 3D FSI example, two structures (a box and a rectangular board) resting on the ground are blown away by a strong wind. From the simulation, we can see the movement/deformation process of the structural parts, as well as the flow vortex formed behind the board.

dCESE-FSI-ibm3D-003: Vacuuming-tube

In this example, three pieces of flimsy material placed on a table are sucked by a vacuuming tube. The pressure inside this vacuuming tube is set to be extremely low, almost at the level of a vacuum, and the vacuuming process and the interaction between the flimsy structural parts and the fluid is simulated.

dCESE-FSI-ibm3D-004: Traffic-light-plate-vibrating

In this 3D FSI example, the deformation and vibration of a traffic light plate due to a strong wind gust is simulated, we can see the interaction process between traffic light plate and wind gust.

dCESE-FSI-ibm3D-005: Blade-rotating

In this 3D FSI-ibm example, a set of blades (tied to the hub) is driven to rotate by a given inlet flow because of the pressure differences between upper and lower blade surfaces.

dCESE-FSI-mmm2D-001: Pole-deformation-mmm2D

This is an example to test our other FSI solver, i.e., the dual CESE FSI-mmm solver which is based on the moving mesh method (mmm). Here, a structural pole (shell element) stands in the middle of a flow field and its bottom end is fixed. When a strong wind flows from left to right, it pushes the pole and forces it to deform. Please note, the FSI-mmm solver is more accurate than the FSI-ibm solver but needs more CPU time because it needs to update the mesh frequently. It is a desirable choice for small deformation FSI problems.

dCESE-FSI-mmm2D-002: Wing-flap-flapping-mmm2D

In this 2D FSI-mmm example, a simple and thin wing-flap (connected to a fixed wing) is used to test flapping in a strong flow field. When the structural deformation is not too large, a moving mesh method FSI solver will be a desirable choice because it is more accurate.

dCESE-FSI-mmm3D-001: Piston-problem-mmm3D

In this example, the flow field inside the piston chamber is simulated during the valve's back and forth motion. Here the dual CESE FSI-mmm solver is used, which is based on the moving mesh method (mmm).

dCESE-2phase-ibm2D-001: Shock-bubble-structure-2D

In this example, the part-I 2D shock-bubble Interaction test case (dCESE-2PHASE-2D-001v) is extended to a FSI test case by adding a thin structure in front of the bubble. We can see how the shock wave interacts with the structure after the shock wave hits the bubble.

dCESE-2phase-ibm3D-001: Shock-bubble-structure-3D

In this example, the previous 3D shock-bubble Interaction test case (dCESE-2phase-3D-001) is extended to a FSI test case by adding a thin structural plate in front of the bubble. We want to see the structure and shock wave interaction more realistically after the shock wave hits the bubble.

dCESE-hybrid-ibm2D-001: Rate-stick-structure-2D

In this FSI example, the previous detonation propagation in a JWL rate stick with confinement in part-I (dCESE-HYBRID-2D-002v) is extended to a fluid/structure interaction (FSI) example by adding a structural part to contain the condensed explosives. And we can see structural deformation under high explosive pressures.

dCESE-hybrid-ibm3D-001: Rate-stick-structure-3D

The above 2D hybrid multiphase FSI example (dCESE-hybrid-ibm2D-001) is extended to a more realistic 3D case to demonstrate our 3D hybrid multiphase FSI solver capability.