New Conjugate-Heat Transfer Solvers in the Compressible CESE Solver in LS-DYNA®

Grant O. Cook, Jr. and Zeng-Chan Zhang
Livermore Software Technology Corp., Livermore, CA 94551

Abstract

Standard coupling methods for performing conjugate-heat transfer between a compressible flow and a structural thermal solver have been previously implemented in the LS-DYNA Conservation Element/ Solution Element (CESE) compressible flow solvers. Unfortunately, they do not conserve energy. One consequence of this in conjugate- heat transfer calculations is that the CESE solver's conservation properties are compromised by the errors in the energy transferred at the interface between the structure and the fluid. And this error subsequently propagates to the interior of the fluid domain. In cases where a steady-state solution is sought, another consequence is that these energy errors may grow in time to the point that an incorrect steady state solution is computed.

While one of the goals of these solvers is avoid non-physical and other erroneous solutions, it needs to be pointed out that the accuracy achieved is still a function of the fidelity of the computational mesh required to resolve the real solution. This is of particular concern in conjugate-heat transfer problems where the fluid boundary layer can be very thin, and large changes in the fluid variables in this fluid boundary layer are typical.

Recently, we have implemented new energy-conservative methods for conjugate heat transfer involving compressible gases in the CESE conjugate-heat transfer solvers. These new solvers have been validated using an analytic test that has an initial singularity, and is thus quite challenging. The new method will be described, and discussed in the context of three types of solvers: 1) a fixed mesh Eulerian CESE solver coupled to a rigid structure, 2) moving mesh CESE FSI solvers, and 3) immersed-boundary method (IBM) CESE FSI solvers [1].

Introduction

Modeling the transfer of energy between coupled fluid and structural domains is important in cooling and heating engineering situations. When the fluid is compressible or the speed of the flow is larger than about 30% of the speed of sound in the fluid, it is necessary to use a compressible flow solver for modeling the fluid. These compressible flow solvers are typically time-explicit in order to permit shock- and other discontinuity-capturing by the method. On the other hand, the structural thermal solver in LS-DYNA is a time-implicit solver. Thus, for conjugate-heat transfer problems with a compressible fluid, the structural thermal solver and the compressible fluid solver each advance in time independently, and exchange data at the fluid-structure interface between time steps.

In situations where the fluid can be accurately treated as incompressible, it is possible for the fluid thermal solver and the structural thermal solver to use the same large time step since each use implicit time stepping. In LS-DYNA, the ICFD incompressible flow solver has a conjugate-heat transfer capability that is monolithically-integrated with the structural thermal solver, and therefore this solver uses the same time step in both the ICFD and structural thermal solvers. Not only can this approach be more efficient than using a compressible flow solver, but it also avoids the consistency issue in the data passed between the solvers at the fluid-structure interface.

However, it must be emphasized that the flow must be essentially incompressible to use this approach.

Standard coupling methods for performing conjugate-heat transfer between a compressible flow and a structural thermal solver have been previously implemented in the LS-DYNA Conservation Element/ Solution Element (CESE) compressible flow solvers [1]. Unfortunately, these standard conjugate-heat transfer methods do not conserve energy. One consequence of this in conjugate-heat transfer calculations is that the CESE solver's conservation properties are compromised by the errors in the energy transferred at the interface between the structure and the fluid. And this error subsequently propagates to the interior of the fluid domain. In cases where a steady-state solution is sought, another consequence is that these energy errors may grow in time to the point that an incorrect steady state solution is computed. Since the high accuracy of CESE methods has been demonstrated to be linked to its local and global conservation properties, it is of great importance to maintain those properties.

Consequently, we have implemented in the CESE conjugate-heat transfer solvers a new energy conservative method for conjugate heat transfer involving compressible gases. This new method is described in the next section. Then an analytic validation test is presented that involves an infinite thick plate in a suddenly accelerated flow. In the subsequent sections, each of the new types of CESE conjugate-heat transfer solver is presented along with some results. There are three types of CESE solvers involved: 1) a fixed mesh Eulerian CESE solver coupled to a rigid structure, 2) moving mesh CESE fluid-structure interaction (FSI) solvers, and 3) immersed-boundary method (IBM) CESE FSI solvers [1].

New Energy-Conservative Conjugate-Heat Transfer Method

In the standard coupling method, the temperature of the solid domain is enforced as a boundary condition at the fluid boundary while the heat flux of the fluid is the net heat flux transferred to the solid.

The new method is similar to other proposals with a thermal Robin-type boundary condition for the solid and a Dirichlet condition for the fluid, but differs in that a corrective step is added. This corrective method allows the cancellation of the possible difference between heat fluxes on the two sides of the interface. This difference arises due to the fluid and structural solvers being decoupled in this case. In [2], it is found that the most stable method involves applying the corrected heat flux only to the solid side of the interface. The general strategy can be written as follows:

$$T^n_{I,F} = \frac{h_S T^0_{I,S} + h_F T^0_{I,F} + \Phi_{I,F} + \Phi^0_{I,S}}{h_S + h_F}, \tag{1}$$

$$\Phi^n_{I,S} \equiv h_S \left(T^n_{I,S} - T^0_{I,F} \right) - \Phi^0_{I,F} + \frac{\Delta(E_S)}{S(\Delta t)}, \tag{2}$$

where subscript 'S' stands for the solid, 'F' stands for the fluid, 'I' represents the fluid-structure interface node, '0' indicates the previous time level values, and 'n' is the advanced time level.

Also, 'T' is a temperature, Φ is a heat flux, 'h' is a heat conductance, 'S' is the interface surface area accounted for at a structural node on the interface, $\Delta(E_S)$ is the energy correction due to the flux mismatch between the fluid and structural solvers, and Δt is the CESE time step..

It should be noted that the analysis and experiments performed in [2] are in the context of a finite volume compressible flow solver coupled with a mechanics solver that also uses an explicit time step, and therefore have to be adjusted slightly to be used with the CESE method solution. The first difference involves all of the CESE variables being located near the cell centroid, and not at the nodes on the interface. The second difference is that CESE invokes the concept of space-time conservation, and hence requires updating of the interface condition at every time step. The techniques described in [2] allow for methods that permit less frequent interface updates. But from our point of view using the CESE method, less frequent updates would be done at the expense of some energy conservation. Clearly, this effect is problem-dependent. A third difference is that the structural and fluid nodes on the fluid-structure interface are not coincident, and thus the fluid variables need to be interpolated to the structural interface nodes. A fourth difference is that the structural thermal solver in our case is taking a large time step since it is an implicit solver. This means that an extra interpolation step needs to be performed to get the structural temperatures and heat fluxes at intermediate times. Since in FSI problems the structure is moving, these interpolations have to take into account changes in the position of the structural nodes and the shape of the elements next to the FSI interface. Another consequence of the implicit structural thermal time step is that the net heat flux (eq. 2) has to be accumulated at the solid nodes on the interface as computed by the new conjugate-heat transfer solver at each CESE time step. These accumulated values are then fed back to the structural thermal solver in its next time step.

Analytic Test Problem

The analytic test case is due to Pozzi [3]. An infinitely thick plate is wetted by a laminar flow that is impulsively accelerated to a constant Mach number of 3. This is a two-dimensional problem, but can also be solved in three-dimensions with non-reflective boundary conditions on the outside boundaries in the third dimension. The solid domain is below the fluid domain, so the fluid-structure (conjugate-heat) interface is on the bottom of the fluid domain and the top of the solid domain. Side boundaries of the fluid domain are treated as periodic, while the top of the fluid domain has an outflow boundary condition. A fixed temperature (900.0) is imposed at the bottom of the structure.

While posed in [2] in non-dimensional terms, we choose to solve this problem in MKS units. The following CESE cards give the initial conditions and fluid properties:

```
*CESE_INITIAL
$      uic       vic       wic     rhoic       pic       tic       hic
 1.04182e3       0.0       0.0            101325.0     300.0
*CESE_MAT_002
$      mid        mu       smu        k0        sk
         7 1.808e-05     110.5    0.0252     194.0
*CESE_EOS_IDEAL_GAS
$    eosid        cv        cp
         5     715.5    1004.5
```

Some important structural solver inputs are:

```
*BOUNDARY_TEMPERATURE_SET
$     nsid      lcid     cmult       loc
         1         0     900.0         0
*INITIAL_TEMPERATURE_SET
$     nsid      temp
         0     300.0
*MAT_THERMAL_ISOTROPIC
$     tmid       tro     tgrlc    tgmult      tlat      hlat
         1     0.000     0.000     0.000     0.000     0.000
$ specific heat / thermal conductivity
$       hc        tc
  8.902e-4    46.363
```

To match with the results in [2], some plots below are given in terms of the following reduced variables:

$$\tau \equiv 1.04182\text{e}3/0.05 * \text{t} \tag{3}$$

$$\theta = (T - 300.0)/300.0 \tag{4}$$

Fixed-mesh CESE Conjugate-heat Transfer

This capability is accessed through a new boundary condition card to the Eulerian CESE solver: *CESE_BOUNDARY_CONJ_HEAT. This card specifies the boundary segments that are on the same surface as the structural thermal boundary, and through which heat can flow.

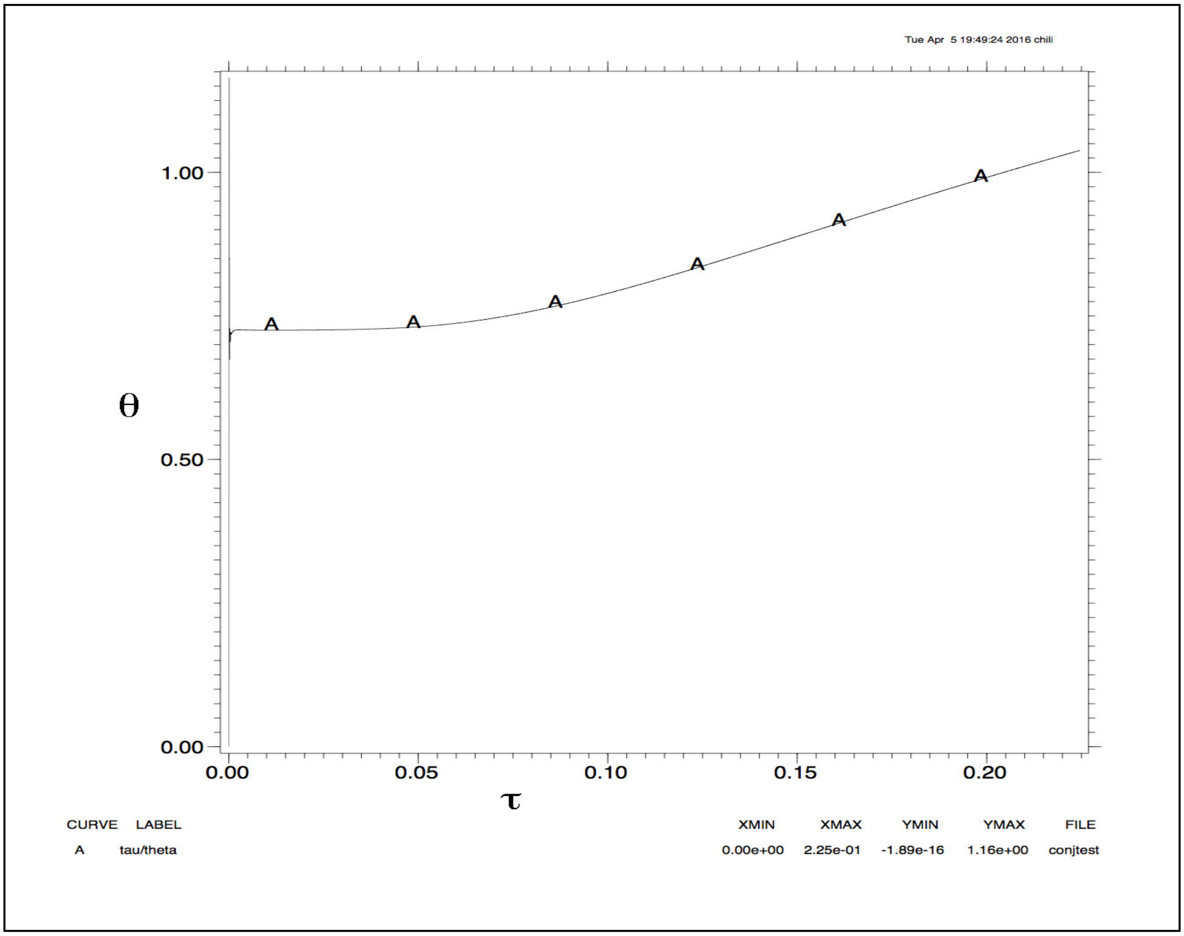

Fig. 1. Typical interface temperature profile for the analytic test problem: relative temperature, θ (eq. 3), as a function of scaled time, τ (eq. 4). Label 'A' shows the minimum and maximum of each variable.

Since the CESE solver in this case is Eulerian, it is assumed that the fluid-structure interface is not moving (no FSI). Fig. 1 shows the validation of this solver for the analytic test problem of the previous section. It should be noted that 2D and 3D versions of each of the solvers discussed in this paper have been validated on this test problem. Another important point is that conjugate-heat transfer only takes place with outside surfaces of solid volume element meshes in 3D and outside mesh faces of shell elements in 2D.

Moving-mesh CESE Conjugate-heat Transfer

In the case of the moving-mesh CESE solvers, it is presumed that an FSI calculation is also taking place. However, a conjugate-heat transfer calculation is only enabled in an FSI calculation if there is also a structural thermal calculation occurring in the same problem. As it is computed internally in the moving mesh CESE solver, the user is not required to explicitly identify the conjugate-heat transfer interface. It is assumed to be coincident with the FSI interface. That is, the FSI part of the problem requires that this interface (both FSI and conjugate-heat transfer) be identified with *CESE_BOUNDARY_FSI card(s). With the moving-mesh CESE solver, a thin boundary layer can be better captured numerically than with the immerse-boundary method CESE solver.

The flow around a mock airfoil with internal cross vents is a test of this solver, although the mesh motion in this case is very small. The incident external flow impinging upon the airfoil is high speed and very hot, while the internal cross flows are close to room temperature and slow. Fig. 2 shows the incident external flow a short time after the arrival of the hot gasses. Fig. 3 shows the internal cooling flows at that same time. Finally, in Fig. 4 is shown the temperature distribution inside the structural solid.

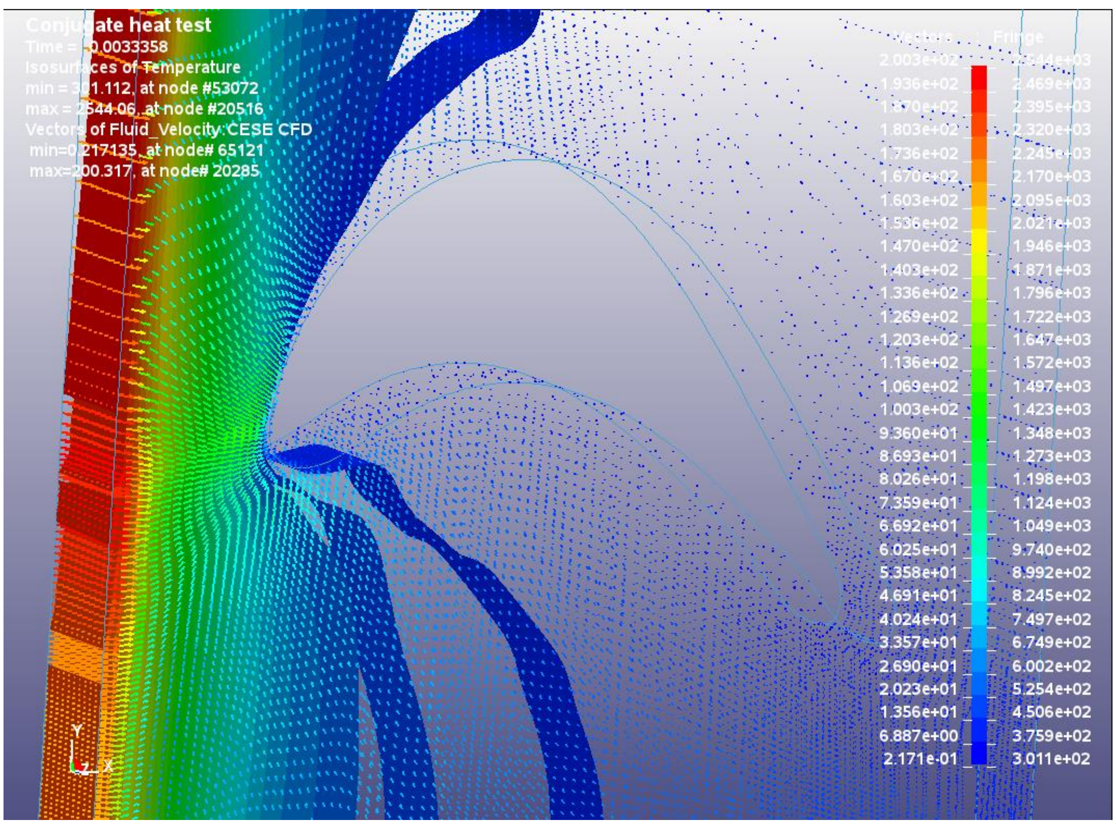

Fig. 2. Isocontours of the external fluid temperature, along with fluid velocity vectors shortly after the arrival of the hot gas.

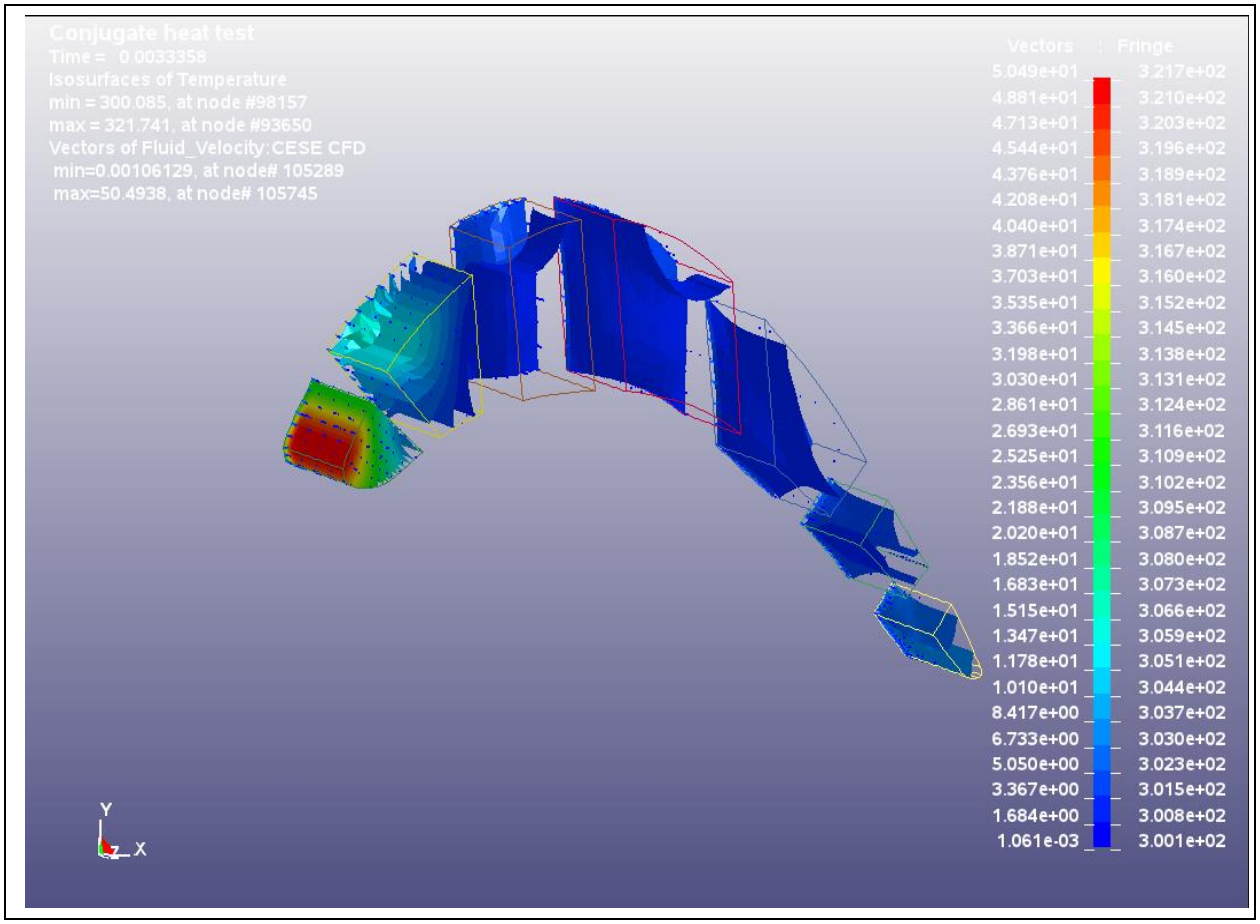

Fig. 3. Isocontours of the internal fluid temperature, along with fluid velocity vectors shortly after the arrival of the hot gas (same time snapshot as in Fig. 2).

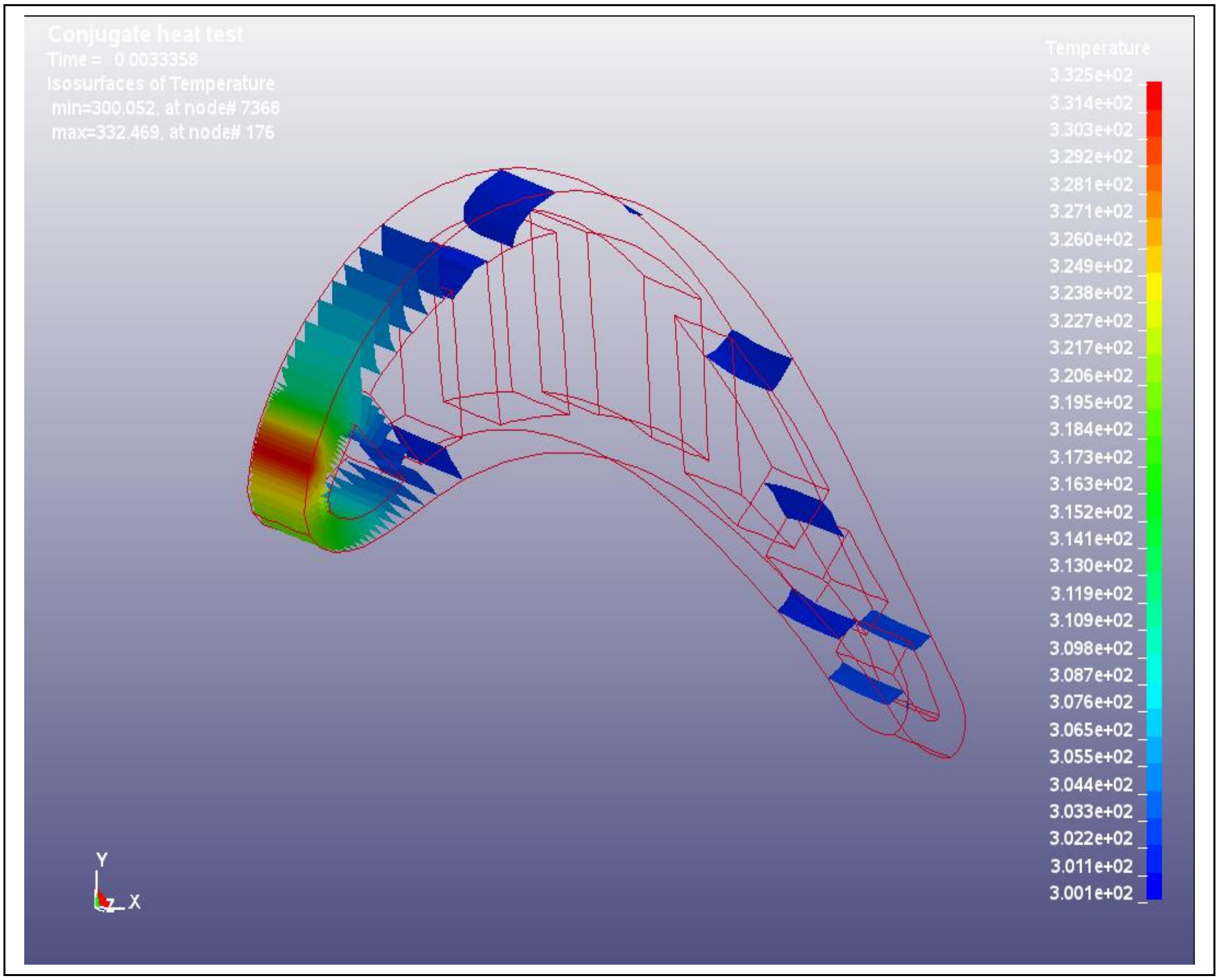

Fig. 4. Isocontours of the structural temperature shortly after the arrival of the hot gas (same time snapshot as in Fig. 2).

New capabilities have been added to d3plot output and to LSPP4.3 to enable viewing the conjugate-heat transfer variables on the fluid-structure interface. In a problem similar to the airfoil above, but without internal cooling, these capabilities are used to view the interface temperature and rate of heat flux transfer to the structure in Fig. 5 and Fig. 6 respectively.

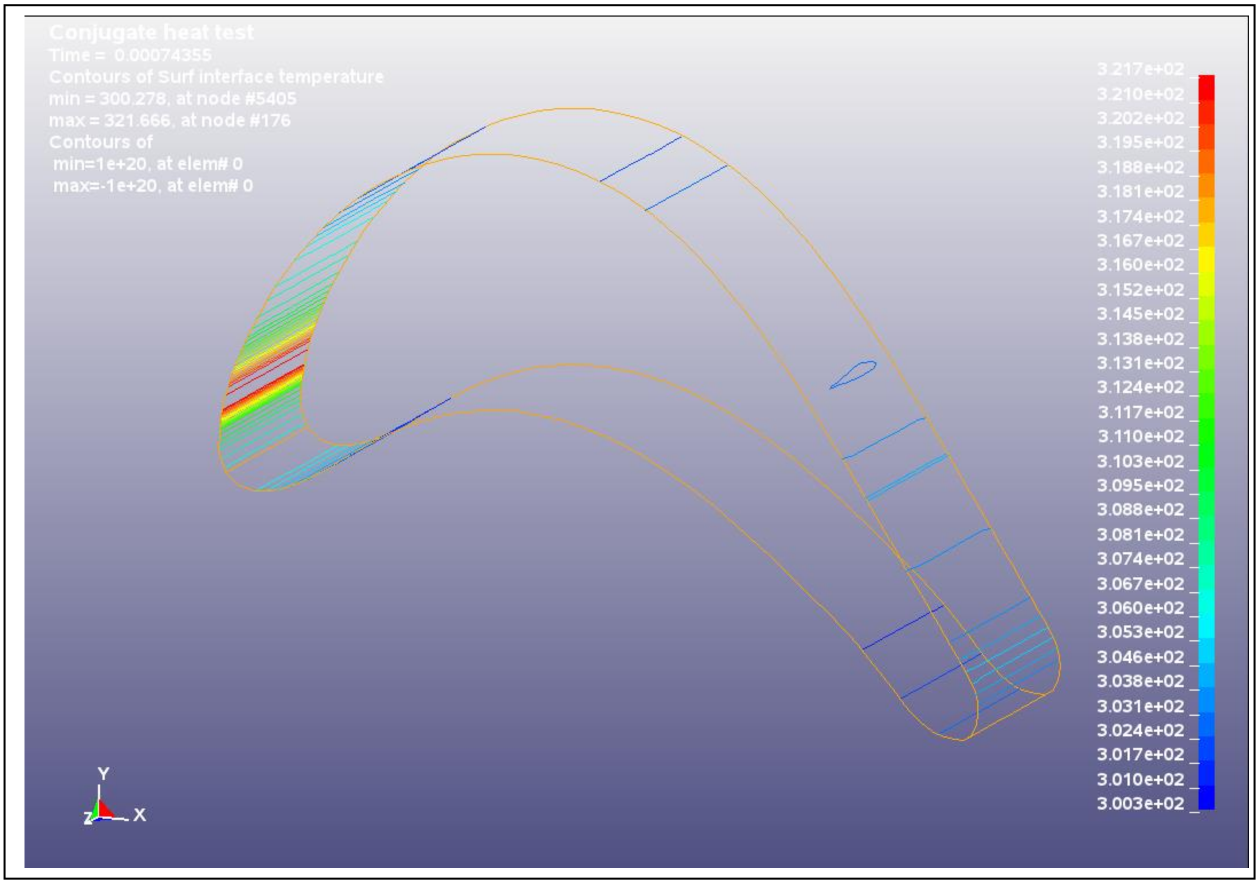

Fig. 5. Interface temperature (eq. 1) found by the new conjugate-heat transfer boundary; simulation with no cooling.

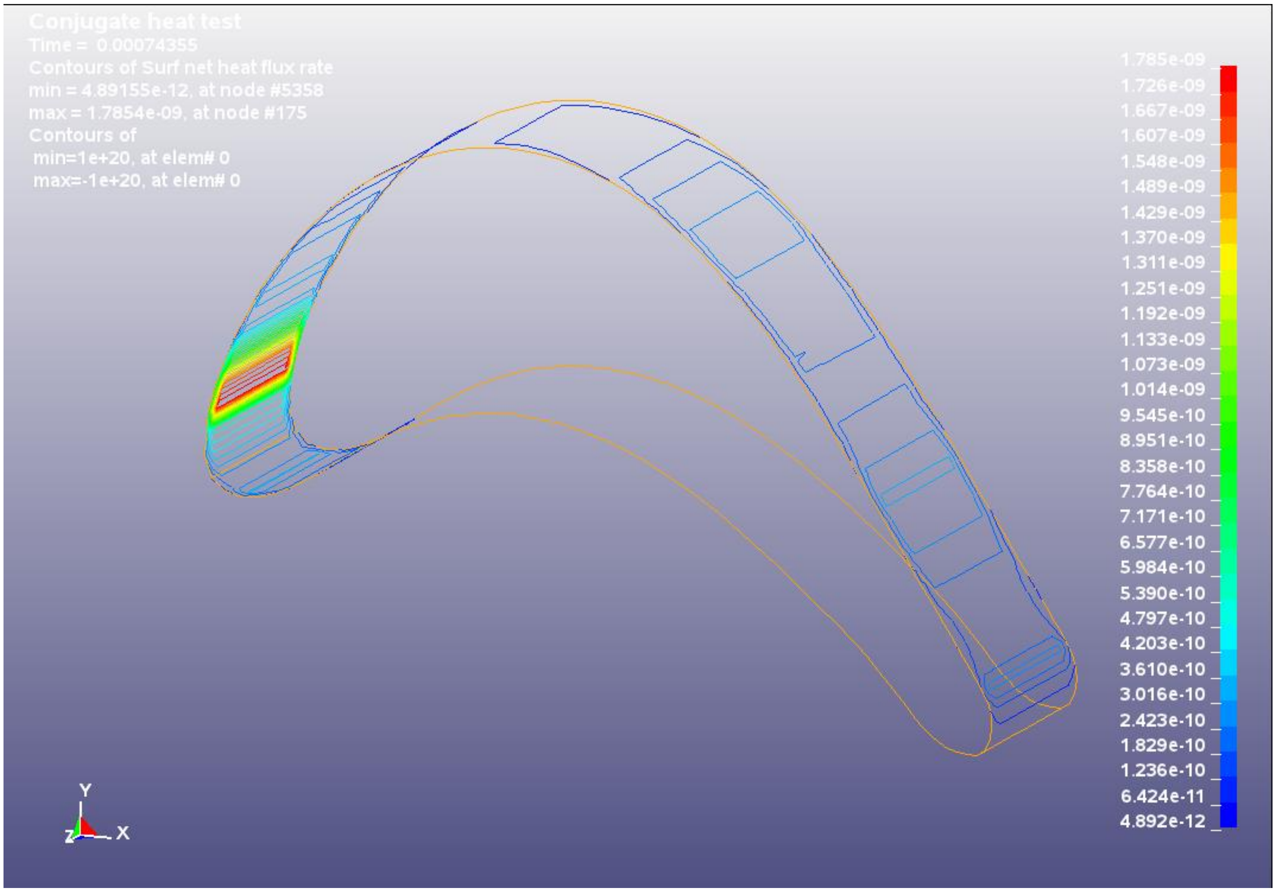

Fig. 6. Instantaneous rate of heat flux transferred to the structure (eq. 2 – per unit time); simulation with no cooling.

Immersed-boundary Method CESE Conjugate-heat Transfer

In the case of the immersed-boundary method CESE solvers, it is once again presumed that an FSI calculation is taking place. And again, a conjugate-heat transfer calculation is only enabled in an FSI calculation if there is also a structural thermal calculation occurring in the same problem. Of all the CESE solvers, this is the most convenient to use. However, developing a CESE mesh that will resolve the fluid boundary layer around the moving structure makes the problem setup for this solver more challenging when rapid changes in the CESE state variables occur in the boundary layer.

Again, the new d3plot capabilities are used to view the heat flux rate of the heat transferred to the structure in a 3D version of the analytic problem with the immersed-boundary solver. Fig. 7 shows the heat flux rate of the heat transferred to the structure. Fig. 8 shows the time history of the heat flux rate at a structural node internal to the conjugate-heat transfer interface. Fig. 9 shows the time history of the conjugate-heat interface temperature at the same structural node. Note that this is the actual temperature, not the relative temperature (eq. 4) used in [2].

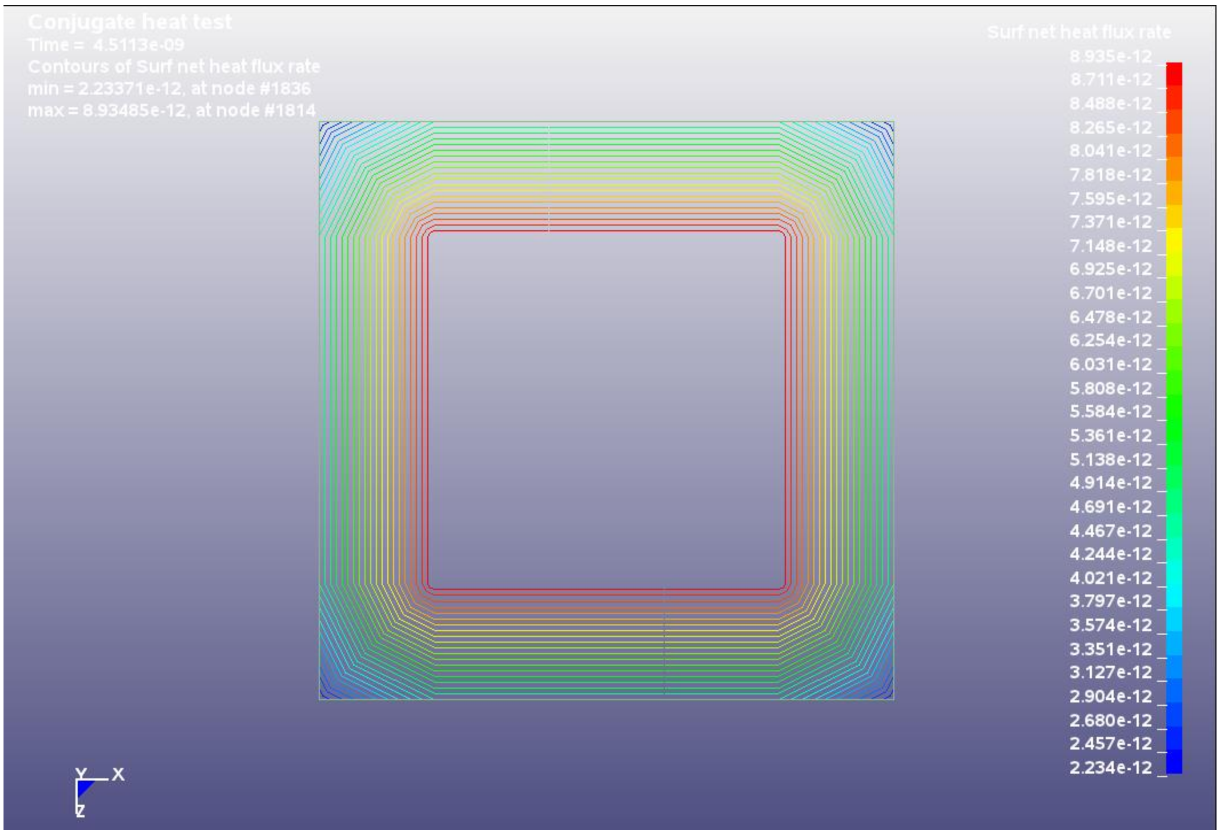

Fig. 7. Instantaneous rate of heat flux transferred to the structure (eq. 2 – per unit time) at early time in a 3D version of the analytic test problem. Shown is the entire conjugate-heat transfer interface plane between the fluid (above) and the structure (below). The immersed-boundary method is used in this case.

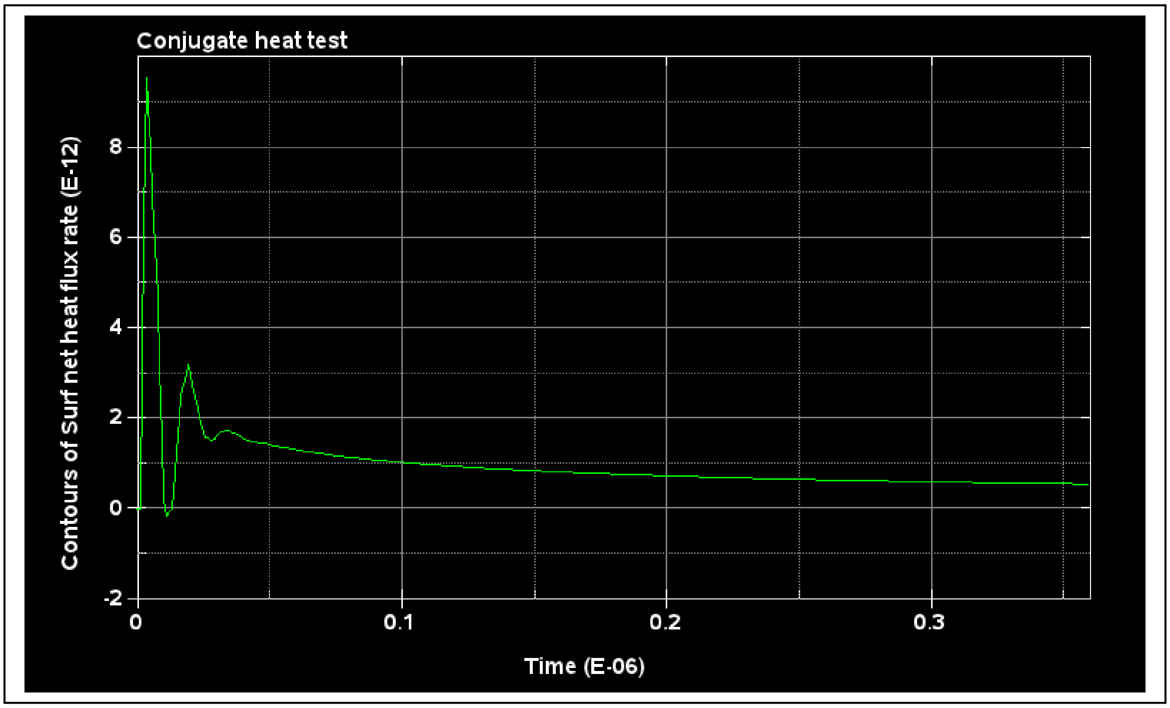

Fig. 8. Time history of the instantaneous rate of heat flux transfer to the solid in the analytic test problem (same test as Fig. 7).

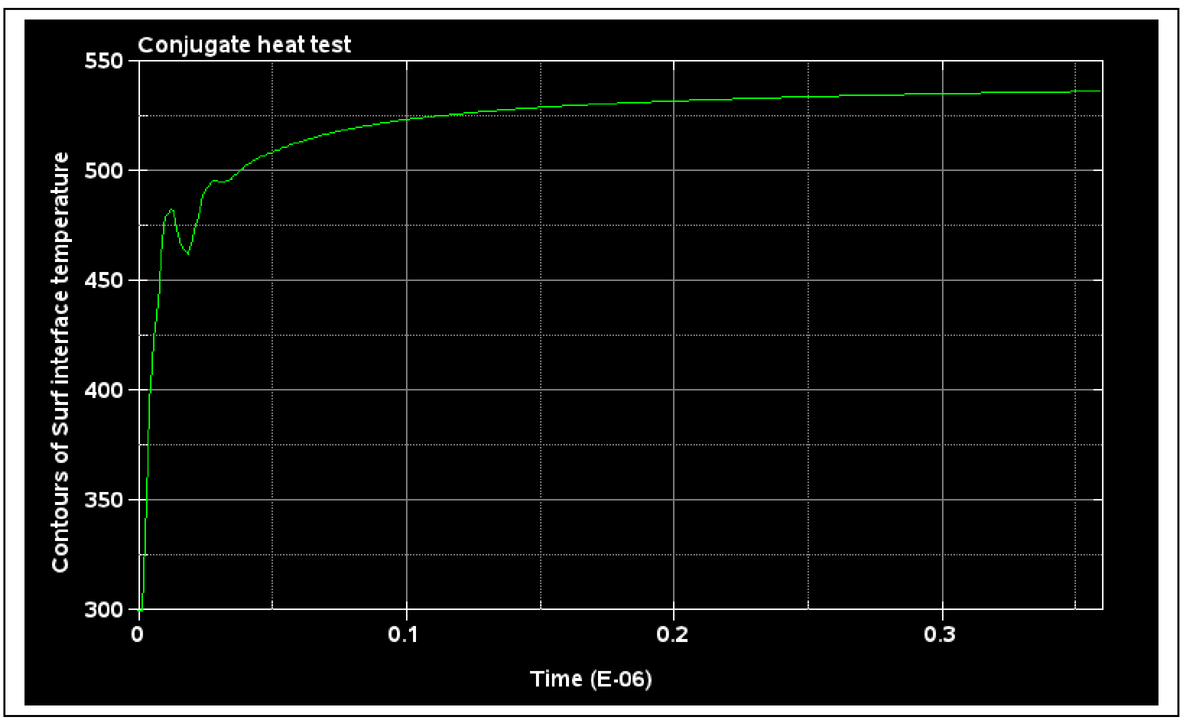

Fig. 9. Time history of the interface temperature (eq. 1) at the same node as in Fig. 8.

Conclusion

With the implementation of the new energy-conservative methods [2], the LS-DYNA CESE solvers now have robust conjugate-heat transfer methods that maintain the conservation properties of these CESE solvers.

These new conjugate-heat transfer solver additions to LS-DYNA CESE solvers have been validated on a challenging analytic test problem in 2D and 3D, and in serial and MPP versions. Testing will continue on more practical tests, and the new d3plot surface output capabilities will aid further verification work.

While one of the goals of the LS-DYNA CESE solvers (and other coupled solvers) is to avoid non-physical and other erroneous solutions, it needs to be pointed out that the accuracy achieved is still a function of the fidelity of the computational mesh required to resolve the real solution. This is of particular concern in conjugate-heat transfer problems where the fluid boundary layer can be very thin, and large changes in the fluid variables in this fluid boundary layer are typical. See the tests in [2] for more details on this sensitivity.

References

1. http://www.lstc.com/applications/cese_cfd/
2. Radenac, E. and Gressier, J. and Millan, P. Methodology of numerical coupling for transient conjugate heat transfer, (2014) Computers & Fluids, vol. 100. pp. 95-107.
3. Pozzi A, Tognaccini R. Time singularities in conjugated thermo-fluid-dynamic phenomena. J Fluid Mech 2005; 538:361–76.