Q&A Session for "Multiphysics Simulation using Implicit Sequential Coupling"

Q: Fluid-Structure object exists for solving fluid/structure interaction in workbench. Why no Fluid-Thermal object for solving thermal/fluid problems in Workbench?

A: Often it is possible to solve thermal-fluid problems entirely within the CFD solution as a conjugate heat transfer problem. As such, the fluid-structure interaction object was the first implementation of the MFX Multi-field Solver in the Workbench Environment.

Q: How do you couple CFD with acoustics? Fluid flowing inside the tube and we are sending sound waves inside it. We need to know how it will reach to other end?

A: CFD can be used to solve aero-acoustic problems, such as noise generated by a flowing fluid. These problems require a very fine mesh and time step in order to capture the sound propagation as a pressure wave in the fluid.

Q: Validation of CFD alone is demanding enough. Can you describe examples of successful validation of coupled flow and deformation models, and also examples where validation was problematic?

A: Validation of coupled flow and deformation models is similar to validating both an individual CFD and FEA model. Mesh refinement and time step refinement studies in conjunction with test data correlation are typically employed to validate a fluid-structure interaction models.

Q: Why is 1-way coupling better than 2-way? Would this be a general statement or just specific to this problem?

A: Stating that 1- or 2-way coupling is better than the other must be done on a per-case basis. In the case of exhaust duct flow the deformations due to pressure fluctuations are small compared to the width or length of the duct. Therefore a 1-way coupling is preferable since the deformations will most likely not affect the flow and the computational time required for 1-way coupling is considerably shorter than for a 2-way coupling.

Q: Have a large Workbench model (115 bodies) w/ a small fluid portion that is expected to gain/lose heat. Getting inaccurate thermal results using the MFX solver to iterate between the small fluid model in CFX and the large solid model in Workbench.

A: There has been a lot of effort put into validating CFX for heat transfer, including the publication of a Heat Transfer Validation report, available free of charge upon request. The first suggestion would be to confirm that best practices for heat transfer predictions for are followed on the fluid side of the mesh.

We normally recommend y+ = ~1, and 10 to 15 nodes in the boundary layer for most accurate heat transfer results. If these conditions are all met, please re-submit this issue to technical support for further investigation.

Q: Which application(s) would you use to model the suction of a piece of paper into a vacuum cleaner? [including the displacement and deformation of the piece of paper as it travels through the hose]

A: Conceivably this problem could be solved with fluid-structure interaction. However, with the large relative motion of the paper in the vacuum hose, remeshing of the CFD domain during the solution would be required. This is a more involved solution procedure versus mesh morphing. Also, the nonlinear transient dynamic response of the paper would involve structural instabilities as the paper folded and wrinkled from the suction of the vacuum. Although this solution is conceivable with FSI, the solution to this problem would most likely stretch the limits of currently available technology.

Q: Ahmad Sereshteh, has anyone modeled/got solution in Hydaulic application with very complex geometry?

A: There are no limitations on the geometric complexity of a model for an fluid-structure interaction solution.

Q: has direct coupling FSI be realized in any known code?

A: Direct coupling for certain classes of fluid-structure interaction problems has been implemented in ANSYS software. We have an element which solves the nonlinear Reynolds Equations for thin film squeeze film damping problems. As far as other codes, I believe ADINA has a fully coupled approach, although I am not familiar with their technology.

Q: Have you tried to setup this problem using workbench?

A: No, but only for historic reasons. At the time when the presented simulations were carried out, Workbench could not handle this. When Workbench 11 was introduced, the FEM specialist still had a long experience from (and preference toward) working with Ansys Prep 7 and we choose to use this interface.

Q: do you have a force versus time curve for the pure cfd analysis on the tank model plotted with the results from the FSI tank model?

A: We do have those results, but they were not shown during the presentation. The force versus time curve from the CFD solution is quite different versus the FSI solution.

____

Q: What does Ansys Prep7 do, in comparison to ANSYS11?

A: Ansys Prep 7 handles scripts and text commands while Ansys 11 is more GUI oriented.

____

Q: With the addition of grid motion, does CFX satisfy the Geometrical Conservation Law, and if so, is it strongly conservative?

A: Yes, an improved implementation introduced in CFX at Release 11.0 uses exact geometrical swept volumes for both transient and advection terms, and is always conservative.

____

Q: Can you review again why the air was modeled as incompressible for the gas exhaust simulation?

A: Since the temperature was 500 deg C, the speed of sound is considerably higher (560 m/s) than the normal 340 m/s at 15 deg C. The criteria for incompressibility is when the fluid velocity is less than 0.3M

____

Q: Why did you cross over the direct coupling approach as not appropriate?

A: Not appropriate since it is not yet feasible for structure (finite element)-fluid (finite volume) calculations. If both domains can be handled by a set of numerical discretisation, direct coupling can be used.

____

Q: What is the timeline on ANSYS + ANSOFT coupling for coupled electro-magnetic-structural simulation?

A: Once the acquisition is finalized, more information on the integration plans will be available.

____

Q: What about the accuracy of the coupling algorithms at the interface? Is it conservative?

A: The algorithm used for load transfer is both profile preserving and conservative.

____

Q: In the FSI problem, does the ANSYS recalculate the pressure caused by the volume change (Mass conservation)?

A: Yes

Q: How do you propose handling problems with impact, which also requires fluid model changes?

A: Highly nonlinear short duration transient fluid-structure interaction problems, such as impact, explosion or bird-strike problems are often solved using explicit dynamics using Euler-Lagrange coupling.

Q: Johan - how do you do the 'sequential' coupling (as you defined it in your presentation). Are you doing a forced response (harmonic) structural analysis using data from the FFT analysis of the CFD pressures?

Q: Was the cluster of PC's for the analysis (last presentation) used at 100% each?

A: The 6 32-bit Windows XP-cluster was used for the CFD, while the FEM analysis was done only on the master node. Not all features in Ansys are parallelizable.

Q: Steve - how was the mesh morphing implimented? Was it in the structural or CFD portion? Which software was used (Ansys? Workbench? CFX? other?)

A: It depends on the coupled solution – for a fluid-structure interaction problem the mesh morphing takes place in CFX, for all other non-structural elements the mesh morphing takes place in ANSYS. In both cases elasticity based mesh morphing is used.

Q: In regard to multiphase flow of gas-solid, what's the best coupling than should be use, pls

A: Both Euler-Euler and Euler-Lagrange methods are available in CFX, either of which may be appropriate, depending on the specifics of the flow and the aims of the simulation.

Q: For 1-way sequential coupling (Fluid-to-Structure), is implicit sequential coupling the best / easiest scheme?

A: For one-way sequential coupling, the coupling is based on a one-way load transfer from the fluid to the structure. This is implemented in the ANSYS Workbench Environment, it is a single mouse click to transfer the fluid pressures as a load imposed on a structural model.

Q: Does mesh pattern and density in two dissimilar meshes affects accuracy allot?

A: The load transfer algorithm is both profile preserving and conservative. A dissimilar mesh interface can be used; however the mesh density needs to be adequate for each individual solution.

Q: Is ANSYS working on ways to reduce that amount of data handling required during these advanced simulations? Even on a stand alone steady-state CFD, we are seeing data files exceeding 1 GB.

A: Fluid-structure interaction solutions do generate a large amount of data, we are always working to improve data structures and file sizes. There are some new options available in version 12.0 for reducing the size of the results files.

Q: Is there any tutorial available for multiphysics modelling?

A: There are tutorials available in the ANSYS documentation. We have also developed a training course for fluid-structure interaction, and multiphysics modeling.

Q: Could ANSYS provide more tutorials for multiphysics analysis in future versions?

A: There are tutorials for multiphysics analysis available in the documentation; we are working to extend the number of tutorials available.

Q: could you please repeat the concept of stagger iterations

A: A stagger iteration is where the implicit coupling of the individual physics solutions takes place. In the case of fluid-structure interaction - the CFD solution sends fluid forces to the structural solution, and the structural solution sends displacements to the CFD solution within each stagger iteration. The stagger iterations are used to achieve the convergence of the interaction between the two solutions by monitoring the rate of change of the fluid forces and the displacements being transferred between the two physics disciplines.

Q: Question for Johan: Do you have to spend a lot of time manually transferring cfx results at all BCs to ansys? Or can this one-way tranfer be automatically handled?

A: The definition of which quantities (pressure, forces, heat transfer coefficient) and which boundaries in each (fluid and solid) domain that are connected have to be made manually. There is also a matter of defining the FEM mesh in surf152 or surf154-elements and exporting the mesh into a .cdb format. Once this is set up the extraction of data for each time step is done automatically by scripting a .cse file.

Q: Is there some rule in which order should the multi-phys. analyses be done? I mean first e.g. thermal and after mechanical?

A: It depends on which physics is driving the solution. If the solid motion were driving the fluid, the structural solution would be first. If the fluid flow was initiating the solid motion the fluid solution would occur first.

Q: What is the timeframe of including all the multiphysics capabilities from classic ANSYS to ANSYS Workbench?

A: With each release of ANSYS additional core solver capabilities migrate to the Workbench Environment. There are a number of significant enhancements that will be released at version 12.0, which will be announced at the upcoming ANSYS conference.

Q: How is the mesh morphing handled and how long does it take to perform the morphing?

A: The mesh morphing is based on a displacement diffusion equation with a mesh stiffness coefficient. The mesh deformation at a surface is diffused into the flow volume interior, with a diffusion coefficient set to be either a function of element size or distance from walls, or any other desired function of the solution or mesh. These options allow the mesh to be relatively stiff in sensitive regions, to avoid mesh folding. Mesh deformation can add between 10% to 50% more CPU time to a calculation, depending on the complexity of the simulation and the mesh.

Q: Please ask to Johan Gullman-Strand (ODS) what is the offshore equipment/component he presented used for.

A: Any gas turbine on a gas/oil production facility is usually connected to either one or several compressors or to a generator. It is the rotating motion of the turbine axis that is used rater than the exhaust gas flow as in the case of an airplane jet engine. The hot exhaust gas can be (and in most cases is) used in a heat exchanger where the thermal energy is transferred to another media.

Q: Could talk you more about acoustic model for far distance propagation coupling with CFX? Is this far field acoustic model can handle sound propagation over different media like air, water.

A: Direct prediction of far-field noise in a CFD code is made somewhat impractical because of the meshing and timestep requirements. First, you must solve the time accurate compressible Navier-Stokes equations. Accurately resolving the propagation of acoustic waves imposes timestep restrictions such that the Courant number is in the range of 1-2 at most. This restriction can be highly costly. Second, the CFD mesh must span all the way to the reception points with enough spatial resolution to directly resolve acoustic waves over the propagation distance with minimal to no numerical damping. These requirements do not make practical sense for many industrial applications. Practical predictions of far-field sound pressure levels are made by first starting with a time accurate CFD calculation of the near field region. Boundary and interior noise sources are taken from the transient CFD calculation, and used as input to an acoustic code using the Lighthill analogy. This method is however a one-way approach that ignores coupling between acoustics and the flow field, and assumes the fluid behaves as an ideal gas.