Overview of Coupling and Multiphysics in Abaqus

1. Introduction
Abaqus has significant capabilities that are used to solve multiphysics problems. These capabilities are developed over many years and fully integrated as core Abaqus functionality, and they have been used extensively for many engineering applications on products and engineering projects in use today. The advantage of Abaqus Multiphysics is the ease with which multiphysics problems can be solved by the Abaqus structural FEA user. From the same model, same element library, same material data, and same load history, an Abaqus structural FEA model can easily be extended to include additional physics interaction. No additional tools, interfaces, or simulation methodology are needed.

2. Types of Coupling
As the name suggests "Coupling" refers to the combined effect of interacting physical phenomenon happening simultaneously in some domain (like the combined effect of heat and structural loads on the distribution of stresses in a jet engine). Multiphysics is a coupled approach in the numerical solution of multiple interacting physical domains. Depending on the type of problem under consideration the interaction between different physical phenomena can be strong or weak. Abaqus is capable to perform: 1) Sequential coupling for weakly coupled problems, and ii) Fully-coupled analysis for strongly coupled problems.

2.1. Sequential Coupling
Some times the interactions can be approximated to be one-way or in other words it is weakly coupled. For instance, the effect of temperature distribution on the deformation of a plate can be simulated in one-way: i) a thermal problem can be solved to find the temperature distribution, and ii) a structural simulation can be performed to find distribution of stresses in the plate subject induced by the loads and thermal strains. Abaqus provides this type of analysis under the name "sequential coupling" and provides the following facilities for this purpose:
- Predefined fields
- Predefined loads

2.2. Fully-coupled
Most of the times, the interaction between different physics is strong and two-way. For example during hot forging process, large plastic deformations can create heat inside a material and cause changes in its mechanical properties (softening) which eventually alters the resistance of the material against the load and cause larger deformations. This type of phenomenon can be simulated in Abaqus using its "Fully-coupled" capabilities.
Abaqus is capable to perform following fully-coupled analyses:
- Thermal-Stress
- Thermal-Electrical
- Thermal-Electrical-Structural
- Piezoelectric
- Eddy Current
- Pore fluid-Stress
- Acoustic-Structural

2.3. Co-simulation
Sometimes one of the physics of interest is not supported by Abaqus. In these situations, it is possible to perform a "co-simulations". The co-simulation technique is a multiphysics capability for run-time

coupling of Abaqus and another analysis program. An Abaqus analysis can be coupled to another Abaqus analysis or to a third-party analysis program to perform multidisciplinary simulations and multidomain (multimodel) coupling.

3. Multiphysics Capabilities in Abaqus
In the following, more details about the capabilities of Abaqus for the multiphysical analysis are provided.

3.1. CFD
Abaqus/CFD provides computational fluid dynamics capabilities with support for preprocessing and postprocessing provided in Abaqus/CAE. Abaqus/CFD can be used to solve nonlinear coupled fluid-thermal and fluid-structural problems on incompressible flows. Following types of problems can be solved:
- Laminar and turbulent: Internal or external flows that are steady-state or transient
- Thermal convective: Problems that involve heat transfer and require an energy equation and that may involve buoyancy-driven flows (i.e., natural convection)
- Deforming-mesh ALE: Abaqus/CFD includes the ability to perform deforming-mesh analyses using an arbitrary Lagrangian Eulerian (ALE) description of the equations of motion, heat transfer, and turbulent transport

3.2. Coupled Eulerian-Lagrangian (CEL)
The Coupled Eulerian-Lagrangian (CEL) in Abaqus provides the ability to simulate a class of problems where the interaction between structures and fluids is important. Such capability can be used in following example applications:
- Gas-flow inflated automotive airbags
- Tire hydroplaning
- Wave loading on offshore structures
- Cosmetic product dispensing
- Fuel tank sloshing
- High-velocity impact and penetration
- Soil-structure interaction, such as excavation and gouging
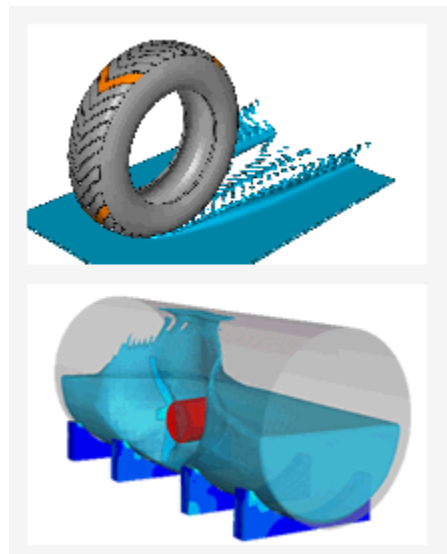
Fig. 1- Samples problems solved by Abaqus that require Coupled Eulerian-Lagrangian capability.

3.3. Hydrostatic-Fluid-Mechanical (HFM)

The Hydrostatic-Fluid-Mechanical (HFM) Multiphysics capability allows the user to include the effects of fully enclosed gas- or liquid-filled cavities in the model. Sample examples that benefit from this feautre include simulating balloons, air bags, seat cushions, athletic shoes, partially filled tanks and other containers, air springs, IV bags, and many other applications that require consideration of the pressure-volume relationship of the enclosure and the energy inside the enclosed fluid. The state equation of the enclosure and the energy in the fluid changes the response of the surrounding structure in significant ways.
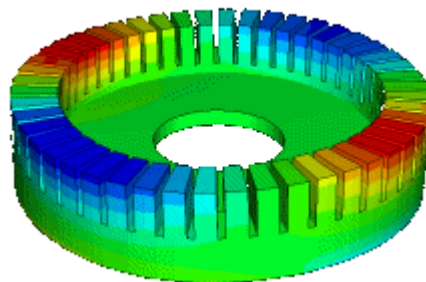
3.4. Structural-Acoustic

Structural-acoustic analysis has diverse application areas including noise transmission, radiation, acoustic attenuation or amplification. Abaqus can perform fully coupled structural-acoustic simulations. Important applications for this functionality include:
- Linear dynamics simulation of acoustic and solid media
- Acoustic finite elements and materials
- Nonreflecting impedance and infinite elements for exterior and radiation problems
- Boolean tools for meshing acoustic cavities
- Fully coupled and uncoupled eigen analysis
- Fully coupled frequency response
- Transient and time-harmonic (steady-state) analysis
- Surface-based fluid-solid coupling
- Acoustics in moving fluids
- Adaptive acoustic meshes for large-deformation enclosures (tires, seal

3.5. Piezoelectric-Mechanical

Piezoelectric-Mechanical has to do with the coupling between structural distortion and electric potential. Some materials exhibit this effect (called piezoelectric or PZT) and electric flow and structural distortions are related in them. Applications for such materials include accelerometers, MEMS devices, medical surgical equipment, control devices.

Abaqus contains a complete two-way electrostatic Piezoelectric-Mechanical simulation capability allowing electric flow to cause straining in the material (and shape change) and also allowing stress to cause electric potential change. The piezoelectric constitutive response in Abaqus is assumed to be linear but piezoelectric elements can be used in both linear and non-linear analyses.

Fig. 2- Abaqus simulation of the stator of an ultrasonic motor. Voltage fluctuations are translated to vertical motion of the stator made from PZT materials.

3.6. Structural-Acoustic

Structural-acoustic analysis has diverse application areas including noise transmission, radiation, acoustic attenuation or amplification. Abaqus can perform fully coupled structural-acoustic simulations. Important applications for this functionality include:

- Linear dynamics simulation of acoustic and solid media
- Acoustic finite elements and materials
- Nonreflecting impedance and infinite elements for exterior and radiation problems
- Boolean tools for meshing acoustic cavities
- Fully coupled and uncoupled eigen analysis
- Fully coupled frequency response
- Transient and time-harmonic (steady-state) analysis
- Surface-based fluid-solid coupling
- Acoustics in moving fluids
- Adaptive acoustic meshes for large-deformation enclosures (tires, seals)

3.7. Thermal-Electric

Thermal-Electric effect (also known as Joule heating) is the interaction between electrical current flow and local resistive heating. In a nutshell, current flow generates heating, heating changes resistivity, resistivity change alters current flow. Abaqus supports this type of simulation which is useful for modeling sensitive electronics devices such as fuses, links, electrical traces, and light bulb filaments. The Thermal-Electric multiphysics capability in Abaqus provides a complete fully-coupled capability including nonlinear effects in material properties in both the electrical and thermal physics components.

3.8. Thermo-Mechanical

Thermal-Mechanical analysis in Abaqus ranges from simple thermal stress (one-way coupling in which an uncoupled heat transfer simulation drives a stress analysis through thermal expansion) to more complex friction-driven heat transfer (in which frictional sliding generates heat as in brake systems) to fully coupled temperature-displacement simulation (in which motion affects heat transfer and heat transfer affects motion).

Examples of application for this module include analysis of the thermo-mechanical durability in many industries from internal combustion engines in automotive to package reliability in electronics. Also, many manufacturing processes such as soldering (cooling from a liquid to solid state), forging, and extrusion involve Thermal-Mechanical multiphysics and require this type of analysis.

Adiabatic solution   Fully-coupled solution
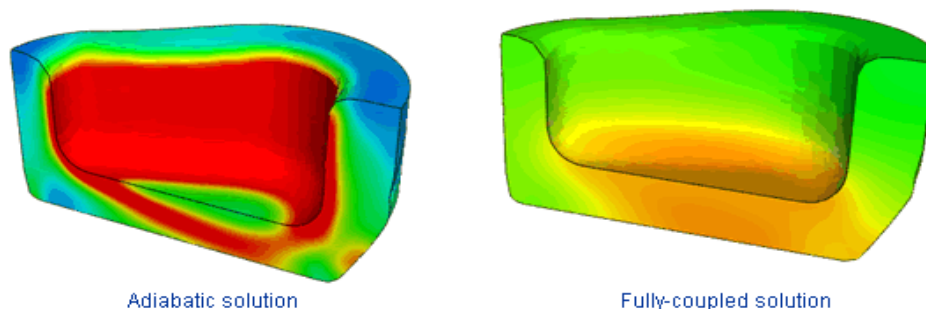
Fig. 3- Thermo-mechanical analysis of a two-stage bulk metal forming process performed in Abaqus. In this problem large plastic strains generate heat and raise the temperature of the part locally and eventually degrade its material properties.

3.9. Thermal-Fluid-Mechanical

This module can be used for the simulation of coupled-thermal, moisture, and mechanical fields. In many industrial processes, the combined effects of moisture and heat on product performance or behavior during assembly can be critical. For example in the packaging of electronics, the module allows for simultaneous consideration of the detailed behavior of a product under realistic operating conditions.

3.10. Structural-Pore Pressure

This module is mostly used for civil engineering simulations when the influence of water on the behavior of soils under load should be simulated. The simulation of the behavior of soils and other natural materials with this module can be used to model dams and tunnels foundations.
This module provides the ability to model:

- Transient or steady-state distribution of fluid pore pressure in soils.
- Consolidation; i.e., the deformation in soils associated with the redistribution of pore water and pressure
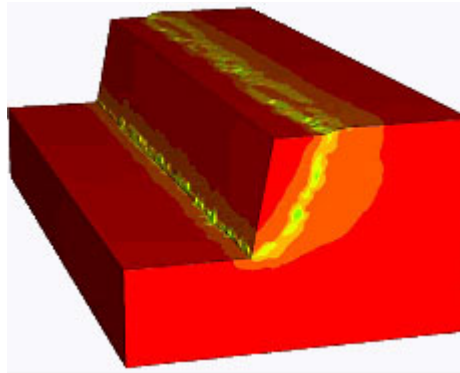
Fig. 4- Application of structural-pore pressure module in the simulation of a dam in Abaqus.

4. More Advanced Topics on Coupling
4.1. Solution Techniques and Convergence

In the thermal-stress analysis in Abaqus/Standard the temperatures are integrated using a backward-difference scheme, and the nonlinear coupled system is solved using Newton's method.
Abaqus/Standard offers an exact as well as an approximate implementation of Newton's method for fully coupled temperature-displacement analysis. The approximate method is computationally cheaper than the exact analysis and requires less memory.
An exact implementation of Newton's method involves a nonsymmetric Jacobian matrix as is illustrated in the following matrix representation of the coupled equations:

$$\begin{bmatrix} K_{uu} & K_{u\theta} \\ K_{\theta u} & K_{\theta\theta} \end{bmatrix} \begin{Bmatrix} \Delta u \\ \Delta\theta \end{Bmatrix} = \begin{Bmatrix} R_u \\ R_\theta \end{Bmatrix},$$

where $\Delta \mathbf{u}$ and $\Delta \boldsymbol{\theta}$ are the respective corrections to the incremental displacement and temperature, $\mathbf{K_{ij}}$ are submatrices of the fully coupled Jacobian matrix, and $\mathbf{R_u}$ and $\mathbf{R_\theta}$ are the mechanical and thermal residual vectors, respectively. Solving this system of equations requires the use of the unsymmetric matrix storage and solution scheme. Furthermore, the mechanical and thermal equations must be solved

simultaneously. The method provides quadratic convergence when the solution estimate is within the radius of convergence of the algorithm.

Some problems require a fully coupled analysis in the sense that the mechanical and thermal solutions evolve simultaneously, but with a weak coupling between the two solutions. In other words, the components in the off-diagonal submatrices $\mathbf{K_{u\theta}}$ are $\mathbf{K_{\theta u}}$ are small compared to the components in the diagonal submatrices. For these problems a less costly solution may be obtained by setting the off-diagonal submatrices to zero so that we obtain an approximate set of equations:

$$\begin{bmatrix} K_{uu} & 0 \\ 0 & K_{\theta\theta} \end{bmatrix} \begin{Bmatrix} \Delta u \\ \Delta\boldsymbol{\theta} \end{Bmatrix} = \begin{Bmatrix} R_u \\ R_\theta \end{Bmatrix}.$$

As a result of this approximation the thermal and mechanical equations can be solved separately, with fewer equations to consider in each subproblem. The savings due to this approximation, measured as solver time per iteration, will be of the order of a factor of two, with similar significant savings in solver storage of the factored stiffness matrix. Further, in many situations the subproblems may be fully symmetric or approximated as symmetric, so that the less costly symmetric storage and solution scheme can be used. The solver time savings for a symmetric solution is an additional factor of two. Unless you explicitly choose the unsymmetric matrix storage and solution scheme, selection of the scheme will depend on other details of the problem.

This modified form of Newton's method does not affect solution accuracy since the fully-coupled effect is considered through the residual vector $\mathbf{R_j}$ at each increment in time. However, the rate of convergence is no longer quadratic and depends strongly on the magnitude of the coupling effect, so more iterations are generally needed to achieve equilibrium than with the exact implementation of Newton's method. When the coupling is significant, the convergence rate becomes very slow and may prohibit obtaining a solution. In such cases the exact implementation of Newton's method is required. In cases where it is possible to use this approximation, the convergence in an increment will depend strongly on the quality of the first guess to the incremental solution

4.2. Steady-State and Transient Analysis

A steady-state coupled temperature-displacement analysis can be performed in Abaqus/Standard. In steady-state cases you should assign an arbitrary "time" scale to the step: you specify a "time" period and "time" incrementation parameters. This time scale is convenient for changing loads and boundary conditions through the step and for obtaining solutions to highly nonlinear (but steady-state) cases; however, for the latter purpose, transient analysis often provides a natural way of coping with the non-linearity. Alternatively, you can perform a transient coupled temperature-displacement analysis. You can control the time incrementation in a transient analysis directly, or Abaqus/Standard can control it automatically. Automatic time incrementation is generally preferred.

4.3.1 Transient Analysis: Automatic/Fixed Time Increment

In coupled temperature-displacement analysis, the time increments can be selected automatically based on a user-prescribed maximum allowable nodal temperature change in an increment, $\Delta\boldsymbol{\theta}_{max}$. Abaqus/Standard will restrict the time increments to ensure that this value is not exceeded at any node (except nodes with boundary conditions) during any increment of the analysis. If you do not specify $\Delta\boldsymbol{\theta}_{max}$, fixed time increments equal to the user-specified initial time increment, $\Delta\mathbf{t_0}$, will be used throughout the analysis.

4.3.2 Spurious Oscillations

In transient analysis with second-order elements there is a relationship between the minimum usable time increment and the element size. A simple guideline is

$$\Delta t > \frac{\rho c}{6k} \Delta \ell^2,$$

where Δt is the time increment, ρ is the density, c is the specific heat, k is the thermal conductivity, and Δl is a typical element dimension (such as the length of a side of an element). If time increments smaller than this value are used in a mesh of second-order elements, spurious oscillations can appear in the solution, in particular in the vicinity of boundaries with rapid temperature changes. These oscillations are nonphysical and may cause problems if temperature-dependent material properties are present. In transient analyses using first-order elements the heat capacity terms are lumped, which eliminates such oscillations but can lead to locally inaccurate solutions for small time increments. If smaller time increments are required, a finer mesh should be used in regions where the temperature changes rapidly. It is worth noting that there is no upper limit on the time increment size (the integration procedure is unconditionally stable) unless nonlinearities cause convergence problems.

4.4. Unit Selection
In coupled problems where two different fields are active, take care when choosing the units of the problem. If the choice of units is such that the terms generated by the equations for each field are different by many orders of magnitude, the precision on some computers may be insufficient to resolve the numerical ill-conditioning of the coupled equations. Therefore, choose units that avoid ill-conditioned matrices. For example, consider using units of MPa instead of Pa (Pascals) for the stress equilibrium equations to reduce the disparity between the magnitudes of the stress equilibrium equations and the heat flux continuity equations.

4.5. Element Selection
Coupled temperature-displacement elements that have both displacements and temperatures as nodal variables are available in both Abaqus/Standard and Abaqus/Explicit. In Abaqus/Standard simultaneous temperature/displacement solution requires the use of such elements; pure displacement elements can be used in part of the model in the fully coupled thermal-stress procedure, but pure heat transfer elements cannot be used. In Abaqus/Explicit any of the available elements can be used in the fully coupled thermal-stress procedure; however, the thermal solution will be obtained only at nodes where the temperature degree of freedom has been activated (i.e., at nodes attached to coupled temperature-displacement elements).

The first-order coupled temperature-displacement elements in Abaqus use a constant temperature over the element to calculate thermal expansion. The second-order coupled temperature-displacement elements in Abaqus/Standard use a lower-order interpolation for temperature than for displacement (parabolic variation of displacements and linear variation of temperature) to obtain a compatible variation of thermal and mechanical strain.

5. References
1- Abaqus 6.13 Documentation
2- http://www.3ds.com/products-services/simulia/products/abaqus/multiphysics/