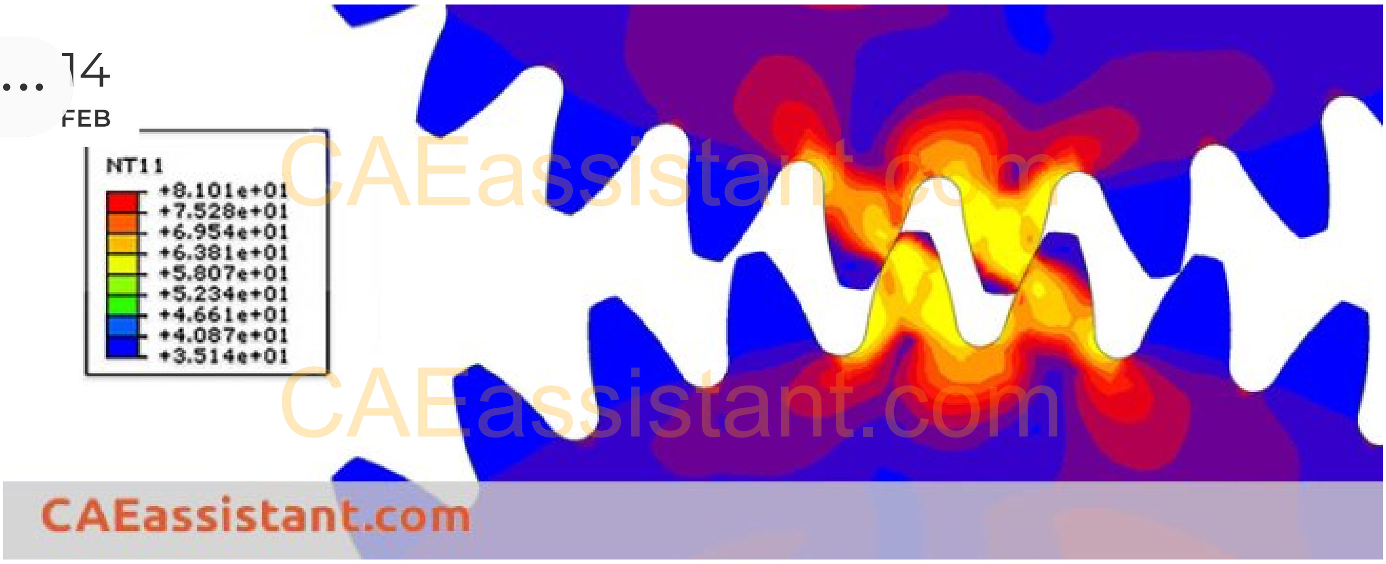
THERMAL ANALYSIS

Heat Transfer and Thermal Stress Analysis in Structural Analysis

Posted by Experts Of CAE Assistant Group

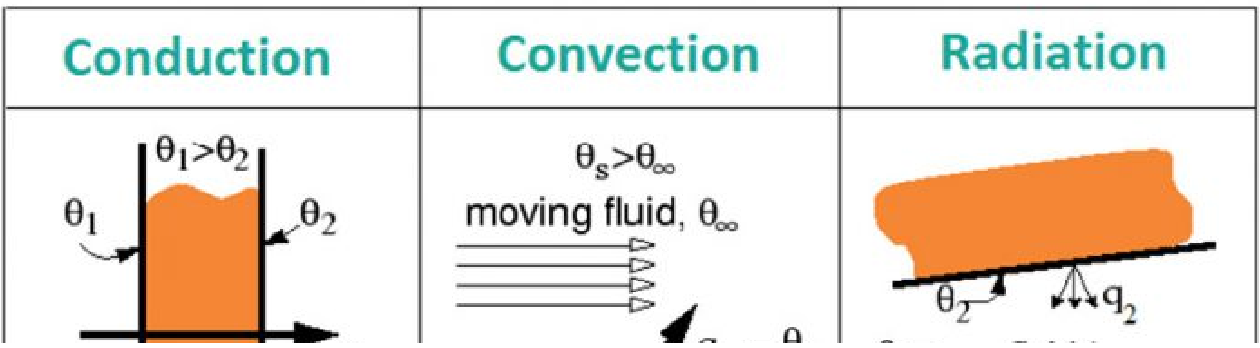
14
FEB

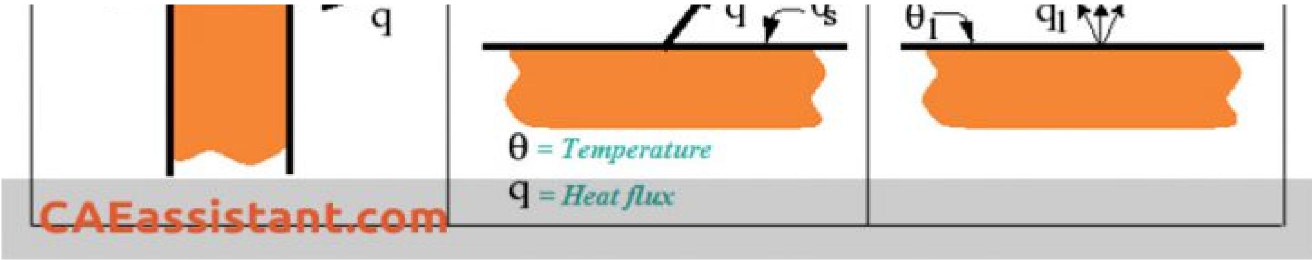


In this post, various Abaqus heat transfer types and thermal stress analyses are discussed, and you can learn heat transfer, thermal-stress analysis with Abaqus in a short and quick way.

1. Modes of Heat Transfer Available in Abaqus

All three modes of heat transfer may be present in an Abaqus analysis:

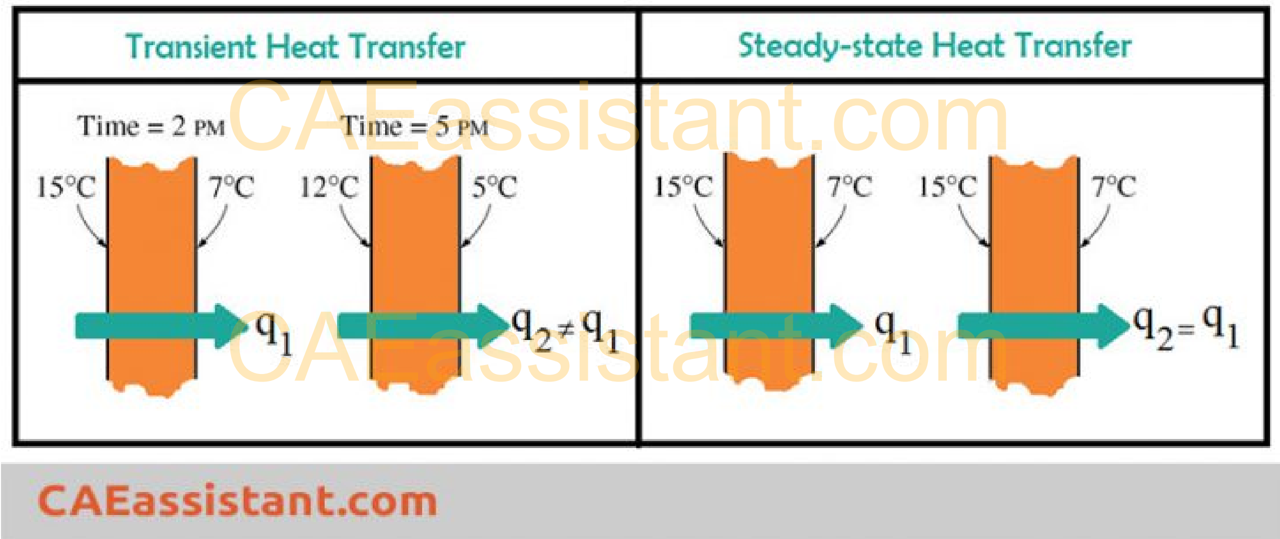




2. Transient and Steady-state Heat Transfer

Heat transfer can be either transient or steady-state:

- Transient: Obtain the response over time with heat capacity effects included.
- Steady-state: Obtain long-term solution under a given set of loads and B.C.s



The term steady implies no change with time at any point within the model, while transient means variation with time or time dependence. Therefore, the temperature or heat flux remains unchanged with time at any location, although both quantities may vary from one location to another.

3. Linear and Non-linear Heat Transfer

Heat transfer problems can be nonlinear because the material properties are temperature-dependent or because the boundary conditions are nonlinear.
Boundary conditions are very often nonlinear; for example, film coefficients can be functions of surface temperature. Radiation effects always make heat transfer problems nonlinear. Nonlinearities in radiation grow as temperatures increase (with power 4 of temperature).
Abaqus/Standard uses an iterative scheme to solve nonlinear heat transfer problems. The scheme uses the Newton method. To be familiar with more linear and non-linear concepts

in the field of heat transfer analysis, you can check the **thermal heat transfer analysis in Abaqus**.

4. Types of heat transfer problems in Abaqus

In general, Abaqus can solve the following types of heat transfer problems (For thermal and thermo-mechanical problems):

- Uncoupled heat transfer analysis
- Sequentially coupled thermal-stress analysis
- Fully coupled thermal-stress analysis
- Adiabatic analysis

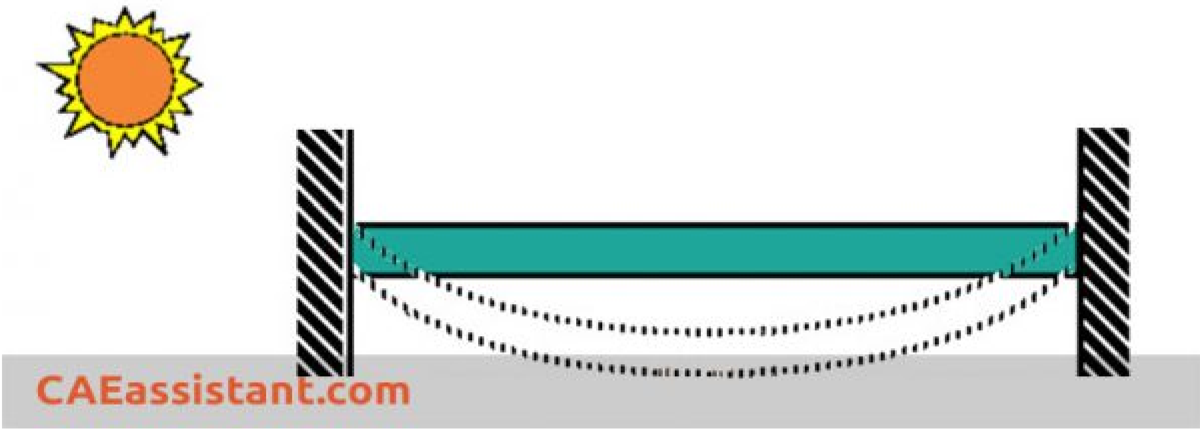
4.1 Uncoupled Heat Transfer

In these analyses, the temperature field is calculated without consideration of the stress/deformation field in the bodies being studied. Pure heat transfer problems may involve conduction, forced convection, and boundary radiation, and can be transient or steady-state and linear or nonlinear. This analysis is just available in Abaqus/Standard.

You can model solid body heat conduction with general, temperature-dependent conductivity, internal energy (including latent heat effects), and quite general convection and radiation boundary conditions. Forced convection of fluid through the mesh can be modeled by using forced convection/diffusion elements.

4.2 Sequentially coupled thermal-stress analysis

When the stress/deformation field in a structure depends on the temperature field, but the temperature field can be found without knowledge of the stress response. It is usually performed by first conducting an uncoupled (pure) heat transfer analysis and then a stress/deformation analysis.



The thermal field affects the mechanical field (for example, Thermal expansion), but the Mechanical field does not affect the thermal field. Besides, mechanical properties can change with temperature.
Two jobs are required to solve the thermal stress problem. Thermal and mechanical fields are solved in sequence (thermal followed by mechanical). The temperatures are read into the stress analysis as a predefined field by reading the output database or the results file from the corresponding heat transfer analysis; the temperature varies with position and is usually time-dependent. It is predefined because it is not changed by the stress analysis solution. However, we have options to apply temperature directly, read it from a file, or be prescribed in a user subroutine.

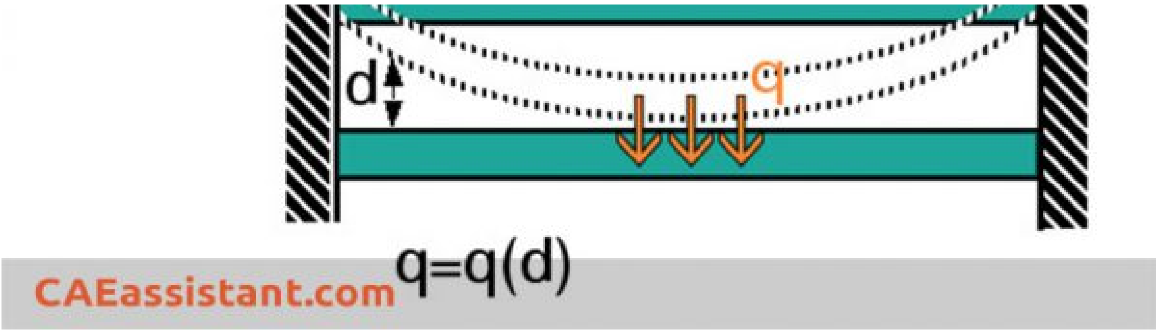
FIG= Predefined Field Window

Sequential thermal-stress analysis in Abaqus/Explicit

To perform a sequential thermal-stress analysis using Abaqus/Explicit, first we run a heat transfer job in Abaqus/Standard. Then we define an Abaqus/Explicit job and Pass temperatures to the Abaqus/Explicit job via the output database or results file.

4.3 Fully coupled thermal-stress

A coupled temperature-displacement procedure (just one job) is used to solve simultaneously for the stress/displacement and the temperature fields. The thermal field affects the mechanical field as above, but here, the mechanical field also can affect the thermal field.

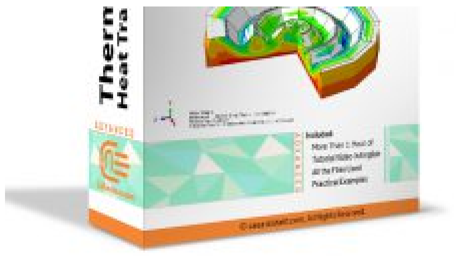


This is used when the thermal and mechanical solutions affect each other strongly: Deformation can change conduction, radiation, etc. and when the heat is generated mechanically due to plastic work or friction. For example, in rapid metalworking problems, the inelastic deformation of the material causes heating, and in contact problems, the heat conducted across gaps may depend strongly on the gap clearance or pressure.
Both Abaqus/Standard and Abaqus/Explicit provide coupled temperature-displacement analysis procedures, but the algorithms used by each program differ considerably. In Abaqus/Standard, the heat transfer problems can be transient or steady-state and linear or nonlinear but Abaqus/Explicit can just model transient thermal (linear or nonlinear) response due to its inherent properties of an explicit integration method.

4.4 Adiabatic Analysis

In cases where mechanical deformation causes heating, but the event is so rapid that this heat has no time to diffuse through the material, we can use adiabatic analysis in Abaqus. This analysis is available in both Abaqus/Standard and Abaqus/Explicit.
An adiabatic analysis is typically used to simulate high-speed manufacturing processes involving large amounts of inelastic strain, where the heating of the material caused by its deformation is an important effect because of temperature-dependent material properties. The temperature increase is calculated directly at the material integration points according to the adiabatic thermal energy increases caused by inelastic deformation; temperature is not a degree of freedom in the problem. No allowance is made for the conduction of heat in an adiabatic analysis.

If you do not have enough information about the different thermal studies yet, the below package helps you understand this concept's theory. This training package includes various workshops to teach in-depth. You can find complete information about thermal analysis in ABAQUS here:



It would be useful to see **Abaqus Documentation** to understand how it would be hard to start an Abaqus simulation without any **Abaqus tutorial**.

NEWER OLDER

LEAVE A REPLY

You must be logged in to post a comment.