## Overview of Loading and Boundary Conditions

### 1.1 General Comments about Loads
External loading can be applied in the following forms:
- Concentrated or distributed tractions.
- Concentrated or distributed fluxes.

Many types of distributed loads are provided; they depend on the element type. This section discusses general concepts that apply to all types of loading. In some situations, concentrated loads and some commonly used distributed loads (such as pressure applied on a surface) may rotate during a geometrically nonlinear analysis. Such loads are known as follower loads.

### 1.2 Types of Loads
There are two ways of specifying distributed loads in Abaqus: element-based distributed loads and surface-based distributed loads. Element-based distributed loads can be prescribed on element bodies, element surfaces, or element edges. Surface-based distributed loads can be prescribed on geometric surfaces or geometric edges. In Abaqus/CAE distributed surface and edge loads can be element-based or surface-based, while distributed body loads are prescribed on geometric bodies or element bodies.

#### 1.2.1 Element-based Loads
Use element-based loads to define distributed loads on element surfaces, element edges, and element bodies. With element-based loads you must provide the element number (or an element set name) and the distributed load type label. The load type label identifies the type of load and the element face or edge on which the load is prescribed. This method of specifying distributed loads is very general and can be used for all distributed load types and elements.

#### 1.2.2 Surface-based Loads
Use surface-based loads to prescribe a distributed load on a geometric surface or geometric edge. With surface-based loads you must specify the surface or edge name and the distributed load type. In Abaqus/CAE surfaces can be defined as collections of geometric faces and edges or collections of element faces and edges.This method of prescribing a distributed load facilitates user input for complex models. It can be used with most element types for which a valid surface can be defined.

### 1.3 Concentrated versus Distributed Loads
#### 1.3.1 Concentrated Loads
In Abaqus/Standard and Abaqus/Explicit analyses concentrated forces or moments can be applied at any nodal degree of freedom. Concentrated loads:
- apply concentrated forces and moments to nodal degrees of freedom; and
- can be fixed in direction; or
- can rotate as the node rotates (referred to as follower forces), resulting in an additional, and possibly unsymmetric, contribution to the load stiffness.

You can specify that the direction of a concentrated force should rotate with the node to which it is applied. This specification should be used only in large-displacement analysis and can be used only at nodes with active rotational degrees of freedom (such as the nodes of beam and shell elements or, in Abaqus/Explicit, tie nodes on a rigid body), excluding the reference node of generalized plane strain elements. If you specify follower forces, the components of the concentrated force must be specified with respect to the reference configuration.

#### 1.3.2 Distributed Loads

Distributed loads:

- can be prescribed on element faces, element bodies, or element edges
- can be prescribed over geometric surfaces or geometric edges
- require that an appropriate distributed load type be specified
- may be of follower type, which can rotate during a geometrically nonlinear analysis and result in an additional (often unsymmetric) contribution to the stiffness matrix that is generally referred to as the load stiffness.

## 1.4 Different Types of Loads

### 1.4.1 Body Forces

Abaqus allows for definition of body forces. Body loads, such as gravity, centrifugal, Coriolis, and rotary acceleration loads, are applied as element-based loads. The units of a body force are force per unit volume.

### 1.4.2 Surface Traction and Pressure Loads

General or shear surface tractions and pressure loads can be applied in Abaqus as element-based or surface-based distributed loads. The units of these loads are force per unit area. In this case Abaqus allows for general/shear surface tractions, pressure, hydrostatic pressure, viscous pressure, stagnation pressure, hydrostatic internal and external pressures (only for pipe and elbow elements).

### 1.4.3 Thermal Loads

Thermal loads can be applied in heat transfer analysis, in fully coupled temperature-displacement analysis, fully coupled thermal-electrical-structural analysis, and in coupled thermal-electrical analysis. The following types of thermal loads are available:

- Concentrated heat flux prescribed at nodes.
- Distributed heat flux prescribed on element faces or surfaces.
- Body heat flux per unit volume.
- Boundary convection defined at nodes, on element faces, or on surfaces.
- Boundary radiation defined at nodes, on element faces, or on surfaces.

### 1.4.4 Electromagnetic Loads

Electromagnetic loads can be applied in Piezoelectric analysis, Coupled thermal-electrical analysis, Fully coupled thermal-electrical-structural analysis, Eddy current analysis, and Magnetostatic analysis. The types of electromagnetic loads available depend on the analysis being performed.

### 1.4.5 Acoustic and Shock Loads

Acoustic loads can be applied only in transient or steady-state dynamic analysis procedures. The following types of acoustic loads are available:

- Boundary impedance defined on element faces or on surfaces.
- Nonreflecting radiation boundaries in exterior problems such as a structure vibrating in an acoustic medium of infinite extent.
- Concentrated pressure-conjugate loads prescribed at acoustic element nodes.
- Temporally and spatially varying pressure loading on acoustic and solid surfaces due to incident waves traveling through the acoustic medium.

### 1.4.6 Pore Fluid Flow

Pore fluid flow can be prescribed in coupled pore fluid diffusion/stress and in the geostatic stress field procedure. Pore fluid flow can be prescribed by:

- defining seepage coefficients and sink pore pressures on element faces or surfaces
- defining drainage-only seepage coefficients on element faces or surfaces that are applied only when surface pore pressures are positive
- prescribing an outward normal flow velocity directly at nodes, on element faces, or on surfaces.

**1.5 Predefined Fields**

Abaqus allows for the definitions of the following types of predefined fields during an analysis:

- temperature
- field variables
- equivalent pressure stress
- mass flow rate

Temperature, field variables, equivalent pressure stress, and mass flow rate are time-dependent, predefined (not solution-dependent) fields that exist over the spatial domain of the model. They can be defined:

- by entering the data directly
- by reading an Abaqus results file generated during a previous analysis (usually an Abaqus/Standard heat transfer analysis)
- in an Abaqus/Standard user subroutine.

Temperature can also be defined by reading an Abaqus output database file generated during a previous analysis. In Abaqus/Standard field variables can also be defined by reading an Abaqus output database file generated during a previous analysis.
Field variables can also be made solution dependent, which allows you to introduce additional nonlinearities in the Abaqus material models.

**2. Boundary Conditions (BCs)**

Almost every finite element analysis requires at least a set of boundary conditions. Boundary conditions:

- can be used to specify the values of all basic solution variables (displacements, rotations, warping amplitude, fluid pressures, pore pressures, temperatures, electrical potentials, normalized concentrations, acoustic pressures, or connector material flow) at nodes
- can be given as “model” input data (within the initial step in Abaqus/CAE) to define zero-valued boundary conditions
- can be given as “history” input data (within an analysis step) to add, modify, or remove zero-valued or nonzero boundary conditions
- can be defined by the user through subroutines DISP for Abaqus/Standard and VDISP for Abaqus/Explicit.

**2.1 Prescribing BCs as Model Data**

Only zero-valued boundary conditions can be prescribed as model data (i.e., in the initial step in Abaqus/CAE). You can specify the data using either “direct” or “type” format. The “type” format is a way of conveniently specifying common types of boundary conditions in stress/displacement analyses. “Direct” format must be used in all other analysis types. The following boundary condition types are available only in Abaqus:

- XSYMM: Symmetry about a plane (X=constant, degrees of freedom 1,5,6 = 0).

- YSYMM: Symmetry about a plane (X=constant, degrees of freedom 1,5,6 = 0).
- ZSYMM: Symmetry about a plane (Z=constant, degrees of freedom 3,4,5 = 0).
- ENCASTRE: Fully built-in (X=constant, degrees of freedom 1,5,6 = 0)
- PINNED: Pinned (degrees of freedom 1,2,3,4,5,6 = 0)
- XASYMM: Antisymmetry about a plane with (X=constant, degrees of freedom 2,3,4 = 0)
- YASYMM: Antisymmetry about a plane with (Y=constant, degrees of freedom 1,3,5 = 0)
- ZASYMM: Antisymmetry about a plane with (Z=constant, degrees of freedom 1,2,6 = 0)

For both “direct” and “type” format you specify the region of the model to which the boundary conditions apply and the degrees of freedom to be restrained. Boundary conditions prescribed as model data can be modified or removed during analysis steps.

**2.2 Prescribing BCs as History Data**

Boundary conditions can be prescribed within an analysis step using either “direct” or “type” format. As with model data boundary conditions, the “type” format can be used only in stress/displacement analyses; whereas, the “direct” format can be used in analysis types. When using the “direct” format, boundary conditions can be defined as the total value of a variable or, in a stress/displacement analysis, as the value of a variable's velocity or acceleration. As many boundary conditions as necessary can be defined in a step.

**2.3 Boundary Condition Propagation**

By default, all boundary conditions defined in the previous general analysis step remain unchanged in the subsequent general step or in subsequent consecutive linear perturbation steps. Boundary conditions do not propagate between linear perturbation steps. You define the boundary conditions in effect for a given step relative to the preexisting boundary conditions. At each new step the existing boundary conditions can be modified and additional boundary conditions can be specified. Alternatively, you can release all previously applied boundary conditions in a step and specify new ones. In this case any boundary conditions that are to be retained must be respecified.

**3. References:**

[1] Abaqus 6.13 Documentation