Appendix to report:

SBJ-33-C5-OON-22-RE-013-APPA

K12 - SHIP IMPACT, GLOBAL ASSESSMENT

Appendix title:

APPENDIX A – ABAQUS TERMINOLOGY

Contract no:	18/91094
Project number:	5187772/12777
Document number:	SBJ-33-C5-OON-22-RE-013-AppA
Date:	15.08.2019
Revision:	0
Number of pages:	5
Prepared by:	Marius Austnes
Controlled by:	Thanh Ngan Nguyen
Approved by:	Kolbjørn Høyland

CONCEPT DEVELOPMENT FLOATING BRIDGE E39 BJØRNAFJORDEN

Table of Content

This report aims to describe the analyses performed as accurately as possible. This necessitates some terminology specific to Abaqus, the FEM analysis software used in this report. Emphasis is on concepts that are specific to Abaqus or terminology that differs from the standard.

A.1 Axes and degrees of freedom

Abaqus numbers its axes 1, 2 and 3. For rectangular coordinate systems (mostly used in this analysis), this corresponds to axes x, y and z. The coordinate system can be global (same as in the global analysis, x axis roughly in the same direction as the girder) or local (used for ship-pontoon and ship-girder connection).

Abaqus labels its translational degrees of freedom U1 (displacement in axis 1), U2 (displacement in axis 2) and U3 (displacement in axis 3). Connecting the translational degrees of freedom of two nodes requires transferring concentrated forces. Rotational degrees of freedom are UR1 (rotation about axis 1), UR2 (rotation about axis 2) and UR3 (rotation about axis 3).

A.2 Connector elements

Elements transfer forces between two or more nodes. For normal elements, the amount of force transferred between two nodes is determined by the element stiffness, which depends on element geometry and element material.
Connector elements are not based on a physical stiffness formulation from geometry and material. Instead, displacement-force-relations can be prescribed directly:

- Degrees of freedom can be forced to be the same, enforcing an infinite stiffness between two nodes.
- Connector elements can behave elastically, so that an arbitrary stiffness matrix can be used
- Connector elements can behave plastically, so that a force-displacement curve can be prescribed.
- Connector elements can include damping, based on the connector displacement rate (velocity).

Connectors can apply to selective degrees of freedom. For example, x-axis forces may be controlled by a plastic force-displacement curve and y displacement rigid.
Connectors have a from-node and a to-node. Connector forces are denoted force from node 1 exerted on node 2.

- **CTF1** [Newton]: Force from node 1 to node 2 in local direction 1
- **CTF2** [Newton]: Force from node 1 to node 2 in local direction 2
- **CTF3** [Newton]: Force from node 1 to node 2 in local direction 3

A.3 Energy measurements

Abaqus provides several energy-related timeseries. These have different acronyms. These measurements are generally over the whole model but may also be recorded for a specific set (specifically noted).

- **ALLKE** [Joule]: Kinetic energy in model integrated over all nodes as a function of time
- **ALLSE** [Joule]: Strain energy in model integrated over all nodes as a function of time
- **ALLVD** [Joule]: Cumulative viscous dissipation in model. Normally monotonically increasing.
- **ALLPD** [Joule]: Cumulative plastic dissipation in model. Normally monotonically increasing.

- **ALLAE** [Joule]: Artificial strain energy. Related to hourglass control and drilling rotation control.
- **ALLWK** [Joule]: Work of the external forces.
- **ALLIE** [Joule]: Total strain energy in model. For the simulations in this report:
  ALLIE = ALLSE + ALLPD + ALLAE
- **ETOAL** [Joule]: Energy balance. For the simulations in this report:
  ETOTAL = ALLKE + ALLIE + ALLFD + ALLVD - ALLWK

A.4 Analysis procedures

Abaqus provides two different procedures for dynamic analyses:

- **Implicit** analyses require dynamic equilibrium in a future time step. This means that changes in displacement and speed for the degrees of freedom depend on future displacement and speed, producing an implicit system of equations, and equation system solution is required.
- **Explicit** analyses require dynamic equilibrium in current and past time steps. Changes in displacement and speed for the degrees of freedom thus only depend on current displacement and speed. We obtain an explicit system of equations that may be solved by regular substitution.

Parameter for the implicit analyses in this report are chosen so that the solution procedure is *unconditionally stable*. This means numeric conservation of energy and momentum. In contrast, explicit analyses are only *conditionally* stable, and generally require very small time increments. CPU time required for solving each time increment, however, is much smaller. This has some general effects on suitability for each solution procedure.

Implicit analyses:

- Can be run interchangeably with static analyses
- Are suited for analyses over a long time frame
- Require smooth nonlinearities

Explicit analyses:

- Cannot be run interchangeably with static analyses
- Are suited for short analyses with detailed time resolution
- Can handle non-smooth nonlinearities

A general rule of thumb is to stick to implicit analyses and bring out Explicit analyses in case of convergence trouble, typically with analyses related to impact, fracture and contact. Impact modeling with use of contact properties in the analysis typically results in non-smooth nonlinearities, which may require explicit analysis. In our case, careful interaction modeling with connectors and smooth force-displacement curves produces a relatively smooth model that the implicit procedure can handle.

1 REFERENCES

[1] Simulia, "Abaqus/CAE 2017," Dassault Systèmes, 2016.

[2] SBJ-33-C5-OON-22-RE-014-B, "Concept developement floating bridge E39 Bjørnafjorden - K12 - Ship impact, Pontoons and columns," Norconsult, Dr. Techn. Olav Olsen, 2019.