University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-WE1

Worked Example 1 using the ABAQUS FE Software

Introduction, Geometry and Initial Mesh

1. Why ABAQUS (not SOLIDWORKS Simulation)?

1.1 Capabilities

Both packages are capable of solving linear stress analysis, determining natural frequencies or linear responses to shock or vibration (relatively) easily and quickly (SOLIDWORKS Simulation actually includes some smart analysis tools that can help make some decisions for you).

However, ABAQUS is a more powerful standalone analysis tool and provides a wider range of material models, element types, boundary conditions and analysis types/options.

ABAQUS is also a flexible analysis tool, allowing users to define their own versions of the above through the use of user-subroutines.

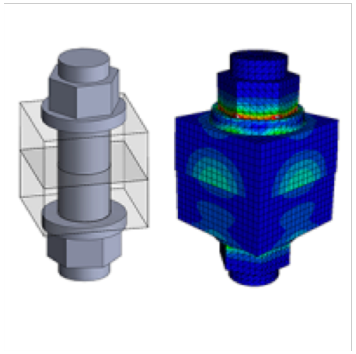
2. Particular analyses where ABAQUS is more appropriate

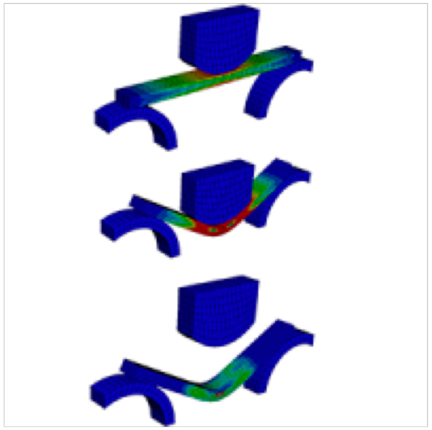
2.1 Non-linear problems

Non-linearities are classified into three main types:

1. Geometric non-linearities – large displacements, large strain, large rotations etc.
2. Material non-linearities – inelastic behaviour or when properties change with loading
3. Contact non-linearities – the area of contact is a function of deformation

Abaqus has a robust contact solver which can deal with multiple contacts in one simulation.



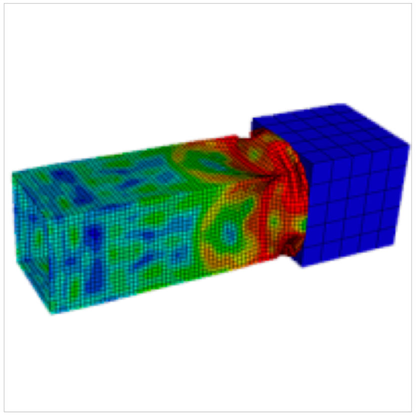


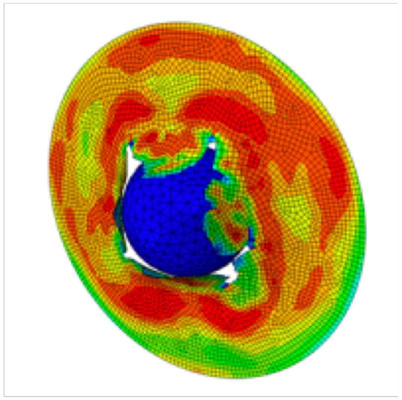
2. Particular analyses where ABAQUS is stronger

2.2 Rapid events

Examples such as shock, drop and impact, especially if they include nonlinear material behaviour.

ABAQUS has an explicit solver (as well as a standard/implicit solver) designed especially for these type of analysis (more robust – less time consuming).





2. Particular analyses where ABAQUS is stronger

2.2 Multiphysics Cases

Real life problems often involve situations including more than one physical phenomena (such as solid mechanics, thermal effects, fluid mechanics, electromagnetics or acoustics…). Sometimes we can decouple the physics to simplify the problem and analyse them separately but there are often situations where we need to couple them in a multiphysics analysis. ABAQUS includes an integrated environment to be able to perform simulations of this type.

3. A brief note on units…

ABAQUS has no inherent set of units, it is the responsibility of the user to make sure that as consistent set of units is used in the analysis.

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2 /in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2 /in^4

4. Introduction

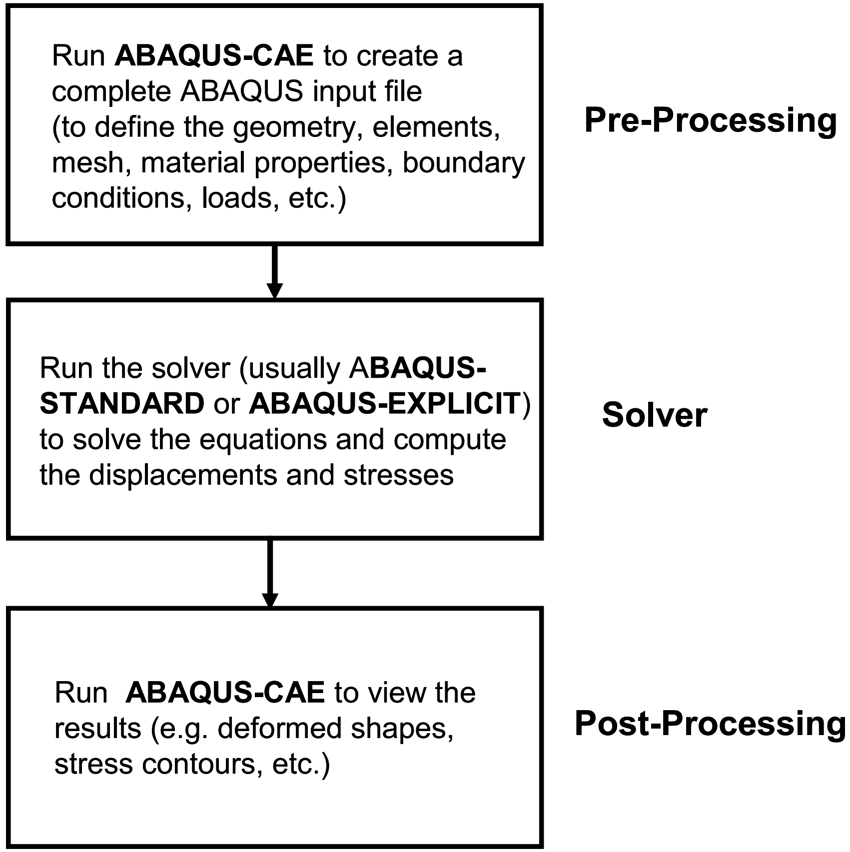


Figure 1: Pre-and post-processing using ABAQUS-CAE

5. Modules in ABAQUS

	Module name	Description
1	**PART**	This module is used to create individual parts by sketching or importing their geometry.
2	**PROPERTY**	This module is used to create section and material definitions and assign them to regions of parts.
3	**ASSEMBLY**	This module is used to create and assemble part instances.
4	**STEP**	This module is used to create and define the analysis steps and associated output requests
5	**INTERACTION**	This module is used to specify the interactions, such as contact, between regions of a model. (This module will not be used in this example)
6	**LOAD**	This module is used to specify loads, displacement boundary conditions, etc.
7	**MESH**	This module is used to create the FE mesh, i.e. generate the elements inside the part.
8	**JOB**	This module is used to submit a job for analysis and monitor its progress.*
9	**VISUALIZATION**	This module is used to view the analysis results, such as stress contours, after a successful run.
10	**SKETCH**	This module is used to create two-dimensional sketches. (This module will not be used in this example)