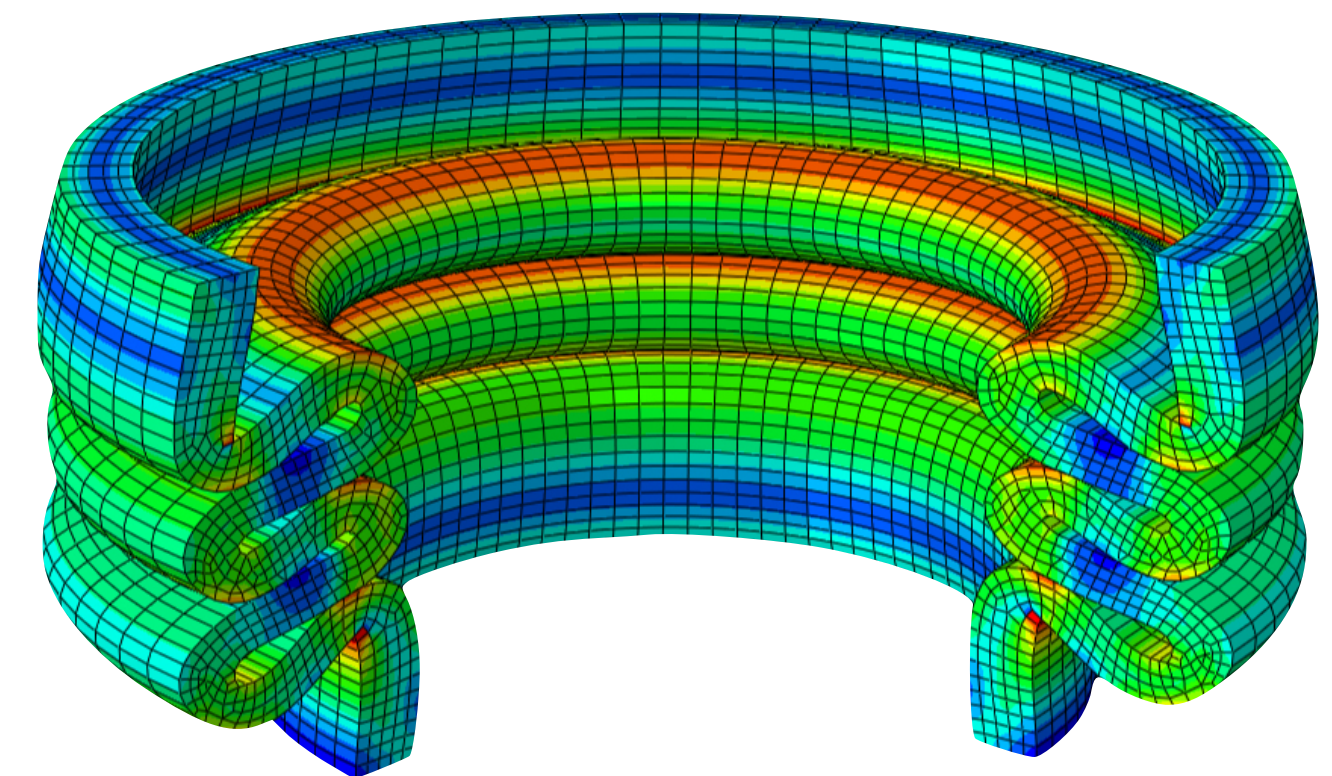# Short introduction about ABAQUS with a step by step example
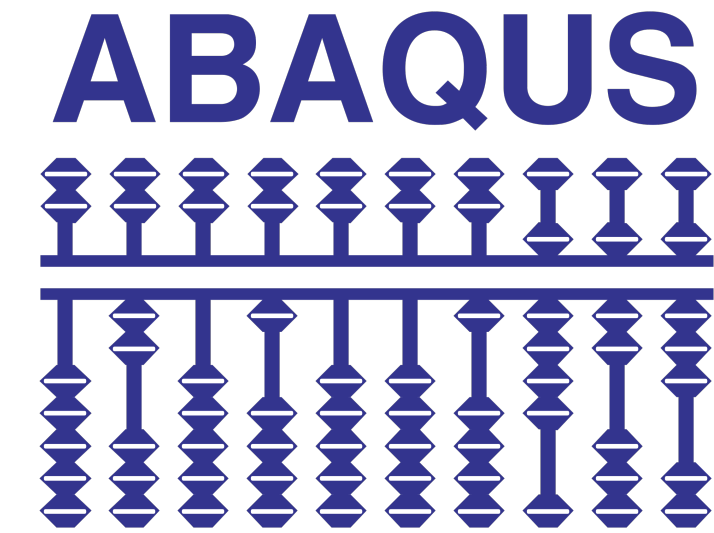
Yangkun DU
Research Associate
yangkun.du@glasgow.ac.uk

28th, Jan 2022

WHAT IS ABAQUS?

ABAQUS

Abaqus is a software suite for finite element analysis and computer-aided engineering, originally released in 1978.

- Abaqus/CAE, or "**C**omplete **A**baqus **E**nvironment". It is a **software application** used for both the modeling and analysis of mechanical components and assemblies (pre-processing) and visualizing the finite element analysis result.
- Abaqus/Standard, a general-purpose Finite-Element **analyzer** that employs implicit integration scheme (traditional).
- Abaqus/Explicit, a special-purpose Finite-Element **analyzer** that employs explicit integration scheme to solve highly nonlinear systems with many complex contacts under transient loads.
- Abaqus/CFD, a Computational Fluid Dynamics **software application** which provides advanced computational fluid dynamics capabilities with extensive support for preprocessing and postprocessing provided in Abaqus/CAE.
- Abaqus/Electromagnetic, a Computational electromagnetics **software application** which solves advanced computational electromagnetic problems.

The Abaqus products use the open-source scripting language Python for scripting and customization.

INSTALLATION AND LEARNING RESOURCES

Abaqus student edition installation instructions: https://edu.3ds.com/sites/default/files/2018-07/Abaqus-SE-2018-InstallationGuide.pdf
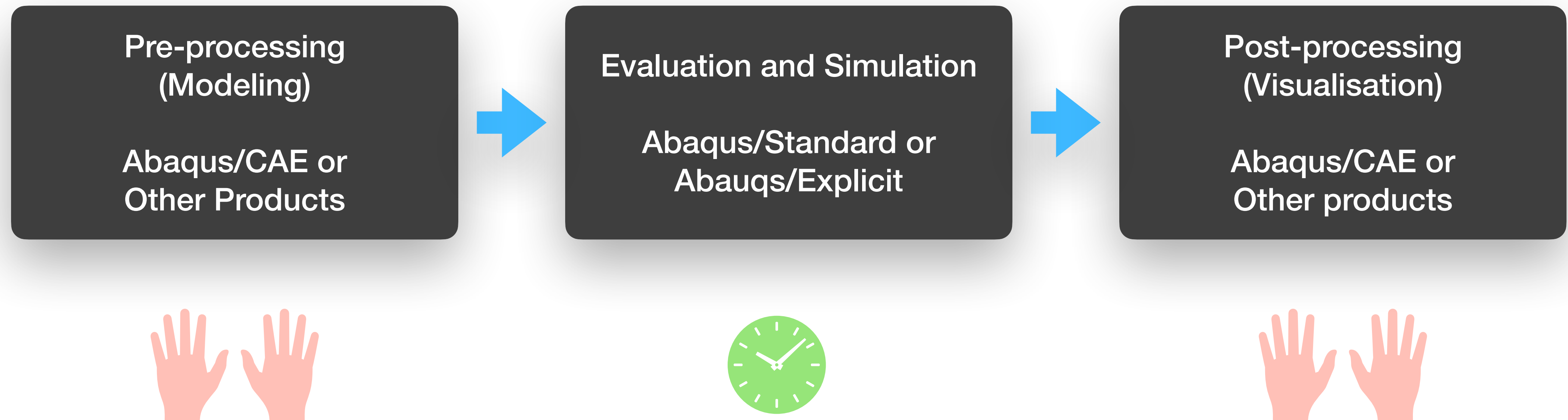
Abaqus student edition is available free of charge to anyone wishing to get started with Abaqus. The Abaqus SE is available on Windows platform only and supports structural models up to 1000 nodes

Official documentation: http://130.149.89.49:2080/v6.14/

Getting started with Abaqus -Tutorial: https://www.youtube.com/watch?v=zdR9mc39KWo

SOLUTION SEQUENCE

Pre-processing
(Modeling)

Abaqus/CAE or
Other Products

Evaluation and Simulation

Abaqus/Standard or
Abauqs/Explicit

Post-processing
(Visualisation)

Abaqus/CAE or
Other products

ABAQUS FILE FORMATS

.cae

Abaqus/CAE model database file contains models and analysis jobs.

.odb

Output database. It is written by the analysis and continue options in Abaqus/Standard and Abaqus/Explicit. It is read by the Visualization module in Abaqus/CAE (Abaqus/Viewer) and by the convert=odb option. This file is required for restart.

.inp

Analysis input file, consisting of a series of lines containing ABAQUS options (keyword lines) and data (data lines).

.jnl

The journal file contains the Abaqus/CAE commands that will replicate the model database that was saved to disk.

WHAT UNITS DOES ABAQUS USE?

- ABAQUS has no inherent set of units.
- It is the User's responsibility to use a consistent set of units.

Table 1. Consistent units.

| 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

COMPONENTS OF THE MAIN WINDOW
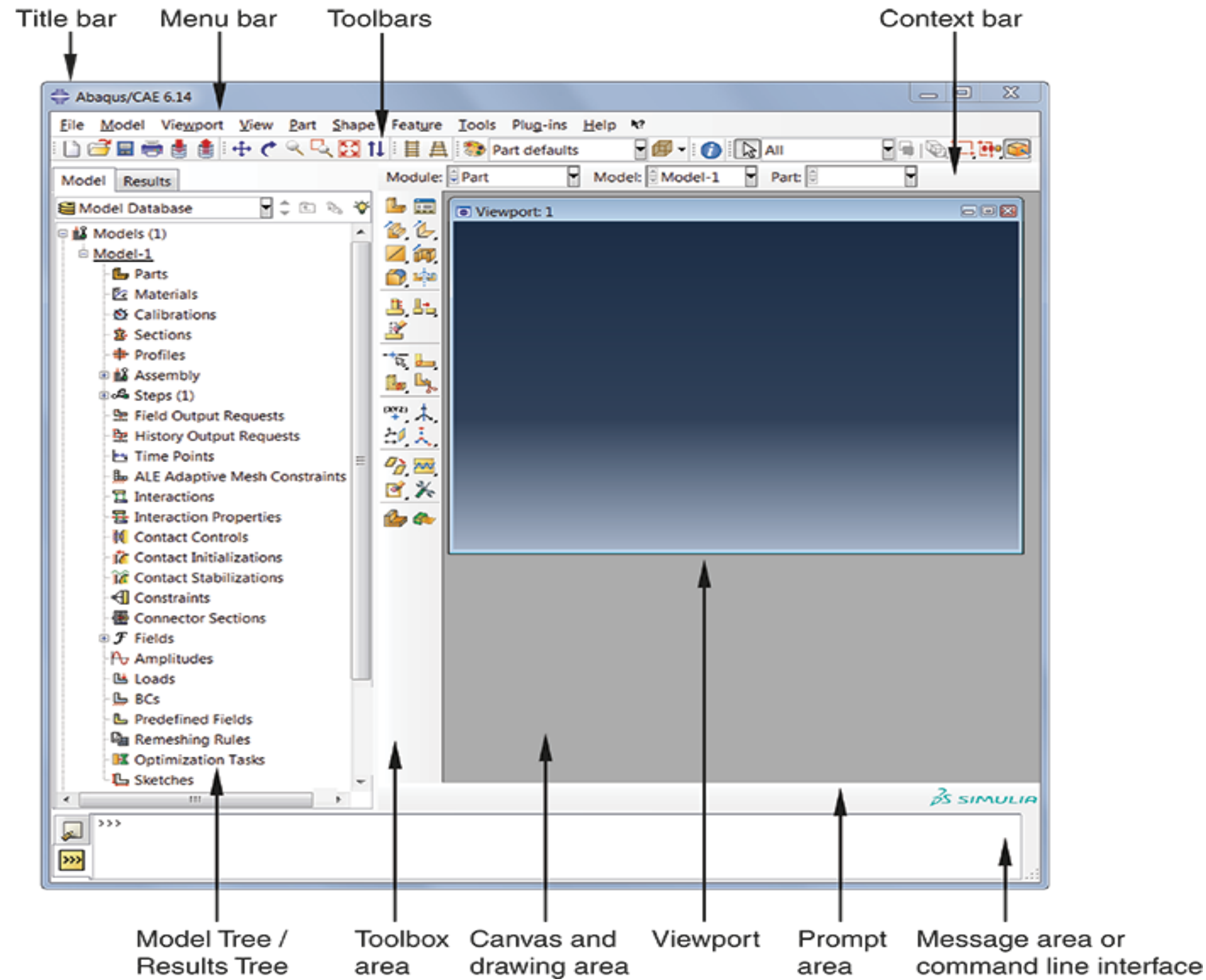
WHAT IS A MODULE?

Abaqus/CAE is divided into functional units called modules. Each module contains only those tools that are relevant to a specific portion of the modeling task.

Part: Create individual parts by sketching or importing their geometry. See Chapter 11, "The Part module."

Property: Create section and material definitions and assign them to regions of parts. See Chapter 12, "The Property module."

Assembly: Create and assemble part instances. See Chapter 13, "The Assembly module."

Step: Create and define the analysis steps and associated output requests. See Chapter 14, "The Step module."

Interaction: Specify the interactions, such as contact, between regions of a model. See Chapter 15, "The Interaction module."

Load: Specify loads, boundary conditions, and fields. See Chapter 16, "The Load module."

Mesh: Create a finite element mesh. See Chapter 17, "The Mesh module."

Optimization: Create and configure an optimization task. See Chapter 18, "The Optimization module."

Job: Submit a job for analysis and monitor its progress. See Chapter 19, "The Job module."

Visualization: View analysis results and selected model data. See Part V, "Viewing results."

Sketch: Create two-dimensional sketches. See Chapter 20, "The Sketch module."

EXAMPLE: FINITE INDENTATION PROBLEM
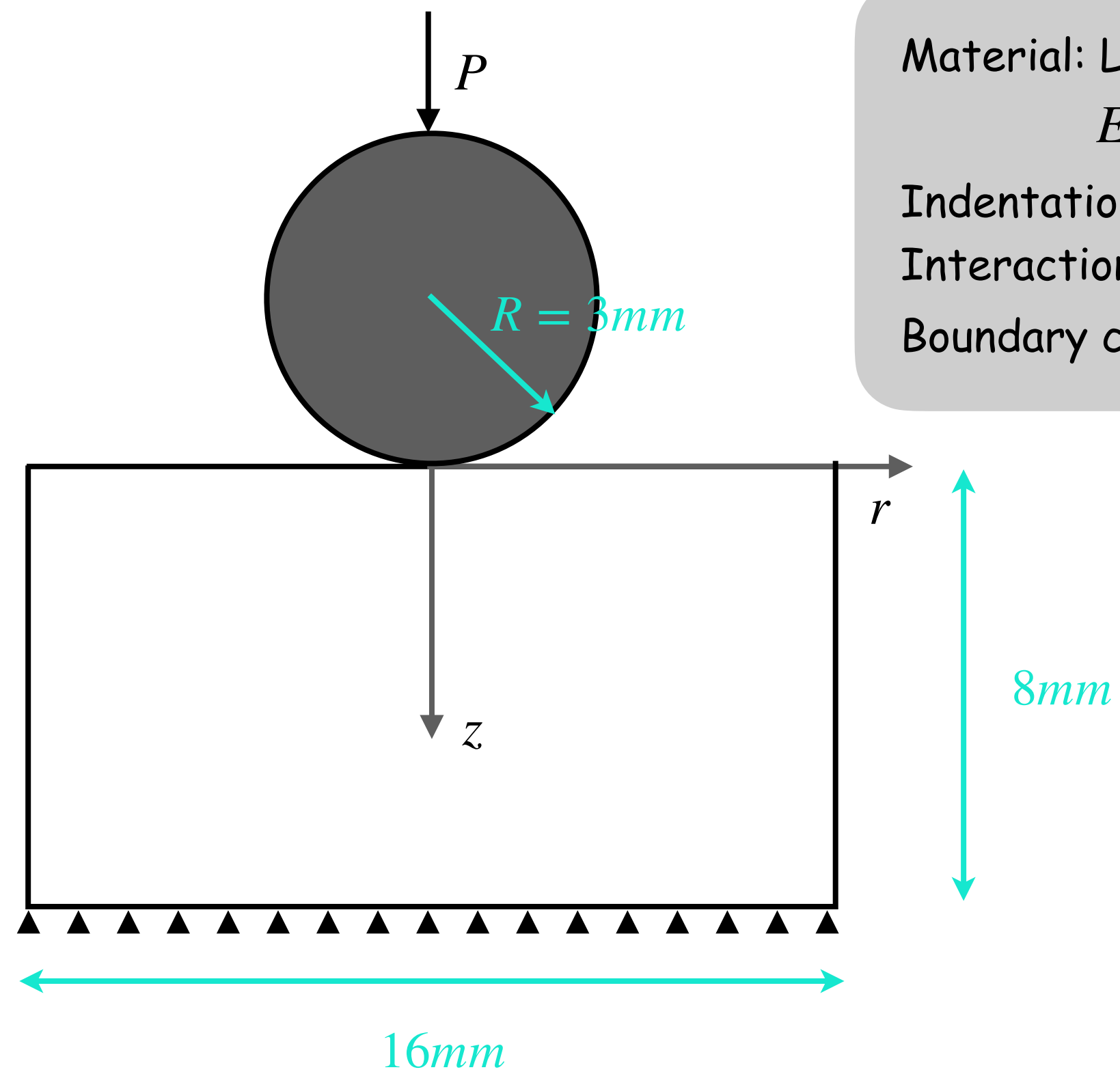
Material: Linear elastic material
$$E = 100kPa, \quad v = 0.3$$
Indentation depth: $D = 1.5cm$
Interaction: Frictionless
Boundary condition: $u_{Bz} = 0$

P

$R = 3mm$

r

z

$8mm$

$16mm$

P

R

D

r

z