



Quiz Questions

Dynamic Analysis using LS-DYNA in Ansys Workbench Part II

Developed and curated by the Ansys part of Synopsys Academic Program

Hesam Moghaddam

education@ansys.com

This resource mentions Ansys LS-DYNA® nonlinear dynamics structural simulation software and Ansys Workbench™ (WB) simulation integration platform.

Below is a series of questions to go along with the Fundamentals of Dynamic Analysis using LS-DYNA in Ansys Workbench Part 2 Lecture.

1. In a force equilibrium chart for quasi-static analysis, what does a close match between applied forces and reaction forces indicate?

- A) Excess inertia effects are present
- B) The system is unstable
- C) The model is maintaining equilibrium
- D) The load is applied too quickly

Answer: C

2. What is a main drawback of using a fully dynamic simulation for a slow-loading problem?

- A) It cannot handle material plasticity
- B) It requires very small time steps, making it computationally expensive
- C) It always produces non-physical results
- D) It eliminates contact interactions

Answer: B

3. When positioning parts in an explicit quasi-static analysis, why should gaps be minimized?

- A) To reduce the natural frequencies of the model
- B) To prevent high-velocity contact closures that introduce dynamic effects
- C) To make the mesh generation easier
- D) To decrease internal energy

Answer: B

© 2026 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found [using this link](#).

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

[Please click the link here](#) out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2026 ANSYS, Inc. All Rights Reserved.