Quiz Questions

Dynamic Analysis using Ansys Workbench LS-DYNA Software

Developed and curated by the Ansys part of Synopsys Academic Program

Hesam Moghaddam

education@ansys.com

© 2026 ANSYS, Inc.

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 Ansys Workbench LS-DYNA Software Part 1 Lecture.

1. What distinguishes structural dynamic analysis from static structural analysis?
    A. It accounts for time-dependent loading and inertia effects
    B. It only uses linear material models
    C. It neglects damping
    D. It ignores deformation

Answer: A

2. In explicit dynamics, the solution is advanced:
    A. Using Newton-Raphson iteration
    B. Implicit time integration
    C. Directly in time without solving a global stiffness matrix
    D. By solving eigenvalue problems

Answer: C

3. Which of the following best describes an appropriate application for Ansys LS-DYNA software in the Ansys Workbench platform?
    A. High-speed impact simulation
    B. Long-duration heat transfer
    C. Static load on a beam
    D. Steady-state fluid flow

Answer: A

4. What is the key feature of the central difference time integration method used in explicit dynamics?
    A. Implicit and unconditionally stable
    B. Conditionally stable with small time steps
    C. Requires matrix inversion at each step
    D. Conditionally stable with large time steps

Answer: B

5. What is the purpose of mass scaling in explicit analysis?
    A. Reduce computational errors
    B. Increase stable time step by artificially increasing mass
    C. Increase mesh density
    D. Reduce memory usage

Answer: B

© 2026 ANSYS, Inc.

6. In a well-behaved explicit dynamics simulation, which of the following should remain approximately constant to indicate stability?
    A. Total Energy
    B. Internal Energy
    C. Hourglass Energy
    D. Kinetic Energy
Answer: A

7. Hourglass control is used in Ansys LS-DYNA software to:
    A. Prevent spurious zero-energy deformation modes
    B. Increase simulation speed
    C. Reduce boundary conditions
    D. Scale mesh quality
Answer: A

8. A high hourglass energy relative to internal energy may indicate:
    A. Model convergence
    B. Valid simulation results
    C. Instability or under-integrated element distortion
    D. Excessive damping
Answer: C

9. In the Taylor impact test, what is primarily analyzed?
    A. Plastic deformation from high-speed impact
    B. Modal frequencies
    C. Structural buckling
    D. Thermal resistance
Answer: A

10. Which of the following is not typically a reason for using explicit analysis over implicit?
    A. Modeling fast transients
    B. Large deformation problems
    C. Long-term creep analysis
    D. Contact-heavy simulations
Answer: C

11. What is the primary objective of mass scaling in explicit analysis?
    A. To improve material damping behavior
    B. To increase the element stiffness
    C. To artificially raise the stable time step by increasing mass
    D. To reduce hourglass modes
Answer: C

© 2026 ANSYS, Inc.

12. The critical time step in an explicit simulation is most strongly dependent on:
    A. Number of elements
    B. Young's modulus
    C. The smallest element length and wave speed (per CFL condition)
    D. Total simulation time
Answer: C

13. According to the Courant-Friedrichs-Lewy (CFL) condition, which of the following helps increase the stable time step in an explicit simulation?
    A. Decreasing material stiffness
    B. Using smaller elements
    C. Increasing wave propagation speed
    D. Reducing density
Answer: A

14. In an ideal explicit dynamic simulation with no energy loss, which of the following should remain approximately constant?
    A. Hourglass energy
    B. Kinetic energy
    C. Total energy (kinetic + internal + contact, etc.)
    D. Work done by boundary conditions
Answer: C

15. A significant increase in hourglass energy during a simulation may indicate:
    A. The simulation is stable and accurate
    B. Excessive artificial stiffness
    C. Numerical instability and possible mesh distortion
    D. Improved accuracy from reduced integration
Answer: C

© 2026 ANSYS, Inc.

© 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.

Ansys 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.
Southepointe
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.