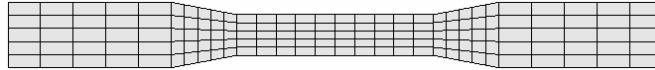


LS-DYNA Implicit Workshop

Problem #1: Tensile Test



Objectives

- Learn how to activate LS-DYNA's implicit mode.
- Learn how to select linear or nonlinear analysis.

Problem Description

A static tensile test is simulated using shell elements. One end of the specimen is constrained, while concentrated nodal loads are applied at the other end. Uniform stresses develop in the narrowed center section.

Input Filename: tensile1.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. Which shell element formulation is used? _____
2. How many steps are used to apply the load? _____

Run the simulation, and post-process using LS-POST. Record below the total applied load and total tip displacement (HINT: the ascii database NODFOR includes data for each of the tip nodes).

3. Applied load _____ Tip displacement _____

Modify the input deck to increase the applied load 100x, and rerun the simulation.
Record the new data:

4. Applied load _____ Tip displacement _____
5. Is the simulation *linear* or *nonlinear*? (Does the tip displacement scale linearly with applied load?)
6. What types of nonlinearity exist in this problem? () material () geometric () contact

Modify the *CONTROL_IMPLICIT_SOLUTION keyword to perform a linear simulation, and record the results below for the large applied load:

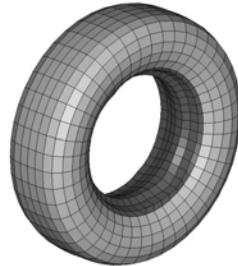
7. Applied load _____ Tip displacement _____
8. What is the reaction force at the fixed end of the beam? _____
9. Why does the reaction force not equal the applied load? _____

Modify the input deck to use the linear element formulation type 21, and repeat the simulation.

10. Does the reaction force now match the applied load? () yes () no

LS-DYNA Implicit Workshop

Problem #2: Car Tire



Objective

- Learn to set up and run implicit dynamic and eigenvalue analyses.
- Learn the advantages of implicit dynamic compared with explicit dynamic analysis.

Problem Description

A coarsely meshed wheel and tire assembly is “kicked” by a brief transient load. The dynamic response of a node at the top of the tire is monitored.

Input Filename: tire.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. What loads and boundary conditions are applied to the tire? _____
-

Run the (explicit) simulation using the original input file. Note the times below.

method	2a. termination time	2b. CPU time
EXPLICIT	seconds	seconds

Copy the input deck to a new directory. Modify the input to run an implicit eigenvalue analysis, computing the lowest 20 eigenvalues. Run the simulation and postprocess the results.

3. What is the lowest frequency? _____ Hz What is it's period? _____ sec.

Copy the input deck to a new directory. Modify the input to run a linear implicit dynamic analysis. Increase the termination time to 1.0 seconds and run the simulation.

method	4a. termination time	4b. CPU time
linear IMPLICIT dynamic	seconds	seconds

Plot the Y-displacement of node ID 1000982 from the NODOUT file, and compare with results from the explicit simulation.

5. Do the explicit and implicit results agree? _____
6. How much faster is the implicit simulation for this (tiny) model? _____
7. How could results of the eigenvalue analysis be used to select the timestep and output interval? _____

Copy the input deck to a new directory. Introduce some Newmark damping (try GAMMA=0.60 and BETA=0.38). Run, plot the Y-displacement of node ID 1000982 from the NODOUT file, and compare with results from the previous simulations.

8. Does the damping most affect high or low frequency response? _____

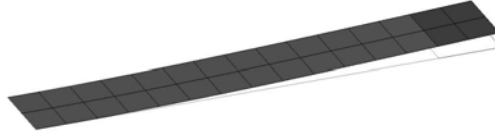
Optional:

Copy the input deck to a new directory, and modify it to perform a **nonlinear** implicit dynamic simulation. Reset the termination time to 0.1 seconds and run the simulation. Compare results for node ID 1000982.

method	8a. termination time	8b. CPU time
n/l IMPLICIT dynamic	seconds	seconds
same, but ISTIF=9999	seconds	seconds

LS-DYNA Implicit Workshop

Problem #3: Rigid Tip



Objective

- Learn the limitations of a linear approximation using a simple example.

Problem Description

A tip load is applied to a cantilevered beam made of shell elements. The tip of the beam is constrained to be rigid.

Input Filename: rigid_tip.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. How is the rigid tip modeled? _____
2. How long is the beam? _____

Run the simulation using the original input file.

3. What type of analysis was performed? _____
4. What is the tip deflection? _____

5. What is the angle of rotation of the rigid body (Note: rotations in the RBDOUT file are given in radians)? _____degrees
6. What is the maximum X-stress at the supported end? _____
7. What is the maximum X-stress in the “rigid” elements? _____

Copy the input file to a new directory. Modify the input file to perform a nonlinear analysis, and re-run the simulation.

8. What is the tip deflection? _____
9. What is the angle of rotation of the rigid body? _____degrees
10. What is the maximum X-stress at the supported end? _____
11. What is the maximum X-stress in the “rigid” elements? _____

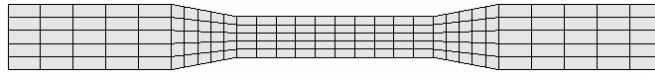
Copy the input file to a new directory. Modify the input file to again perform a linear analysis, but with the constrained nodal rigid body removed. Run the simulation.

12. What is the tip deflection? _____
13. What is the maximum X-stress at the supported end? _____
14. What is the maximum X-stress in the tip elements? _____

Why does stress develop in the “rigid” elements during the linear analysis? (Hint: plot the (axial) X-displacement along the length of the beam for each simulation.)

LS-DYNA Implicit Workshop

Problem #4: Elastic-Plastic Tensile Test



Objectives

- Learn how to observe convergence behavior of nonlinear equilibrium iterations.
- Learn to use automatic time step control for nonlinear problems.
- Learn the benefits of force vs. displacement controlled simulations.

Problem Description

A static tensile test is simulated using shell elements and a nonlinear, elastic-plastic material model. One end of the specimen is constrained, while concentrated nodal loads are applied at the other end. Uniform stresses develop in the narrowed center section.

Input Filename: tensile2.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. Which material model is used? What is the yield stress? _____
2. How is load applied? _____
3. How many steps are used to apply the load? _____

Run the input deck. Does the job run to completion? () normal termination () error termination

4. At what time does the solution begin to struggle? time = _____

Using the postprocessor, plot the X-displacement of an end node vs. time.

5. What is happening at the time shown above? _____

6. max end displacement _____ max eff. stress _____ max eff. strain _____

Activate the nonlinear print flag to get more information about the nonlinear solution process, and repeat the simulation.

7. What two methods are available for this? _____

8. How many cycles are used in the simulation? _____

Switch from load control to displacement control, and repeat the simulation (Hint: helpful keywords are commented out in the original input deck). Using the postprocessor, again plot the X-displacement of an end node.

9. max end displacement _____ max eff. stress _____ max eff. strain _____

10. Why is this problem easier to solve? _____

Return to the original input deck, and activate automatic time step control (IAUTO=1 on the keyword *CONTROL_IMPLICIT_AUTO). Use 200 as the optimum iteration count, and set the maximum stepsize to 0.050. Repeat the simulation.

11. What happens? _____

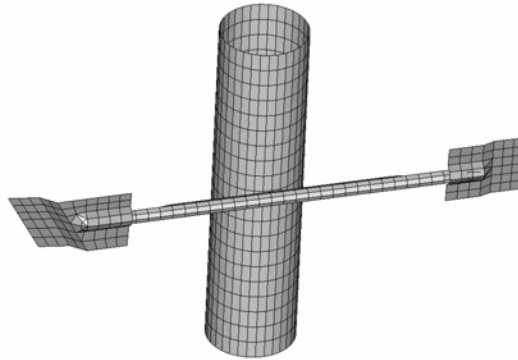
12. max end displacement _____ max eff. stress _____ max eff. strain _____

Using the “ASCII” menu, load the GLSTAT database and plot the step size vs. time.

13. When does the step size change? Why? _____

LS-DYNA Implicit Workshop

Problem #5: Door Beam



Objective

- Learn to perform a springback simulation using an entirely static implicit analysis.
- Learn to specify key points during the solution which must be reached exactly.
- Learn about the available displacement convergence norm options.

Problem Description

A doorbeam subassembly is deformed by a rigid pole. Shell elements are used throughout, and nodal rigid bodies are used to spotweld the components of the doorbeam. The pole is displaced to deform the doorbeam, then retracted to evaluate springback.

Input Filename: doorbeam.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. What type of contact interface is selected? Why? _____
2. Why is the doorbeam chosen as the slave side? _____

Run the simulation and observe the convergence behavior.

Activate the flag IGAPF=2 on optional card "C" in the *CONTACT_... keyword.

Repeat the simulation.

3. Which gap flag produces the best convergence behavior? _____

Postprocess the results, and plot the slave interface force. Save the interface force curve as file "curve1".

4. What is the strategy for solving the springback problem? How is the applied load removed? _____

5. At what time is the full load applied? (HINT: Check the input file.) _____

6. Why are the maximum reaction force and springback predictions from this simulation misleading? _____

Define a key point at time=1.0 using the *CONTROL_IMPLICIT_AUTO keyword. Use a maximum step size of 0.05 throughout the entire simulation. Repeat the simulation, and plot the new interface force curve. Save this curve as file "curve2". Compare these results to "curve1".

7. Is the peak load computed at the correct time? _____

8. Why is the force-deflection behavior not smooth? _____

Select the alternate displacement tolerance scheme using DNORM=1 on *CONTROL_IMPLICIT_SOLUTION, and repeat the simulation.

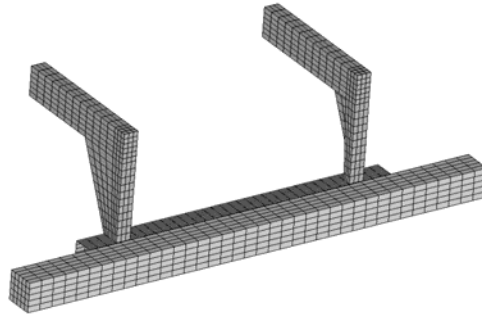
9. Does the alternative displacement tolerance become () more or () less strict when total displacements are large, as they are at the end of this problem?

10. Is the force-deflection curve more smooth? Why? _____

11. What is the springback deflection at the center of the doorbeam? _____

LS-DYNA Implicit Workshop

Problem #6: Truck Bumper



Objectives

- Learn the behavior of contact interfaces in static implicit simulations.
- Learn how to set input parameters for implicit contact problems.

Problem Description

The rear bumper of a truck is modeled using shell elements. A solid, rigid bar is displaced into the bumper, causing plastic buckling of the support. Post-buckling response is determined.

Input Filename: bumper.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. What type of contact interface is used? _____
2. Is the thickness of the shell elements considered? _____

Run the simulation and postprocess the results.

3. Why does the simulation stop? _____

Temporarily increase the default displacement convergence tolerance to DCTOL=0.005 using keyword *CONTROL_IMPLICIT_SOLUTION, and repeat the simulation. Stop the job after about 10 steps using the interactive command “<ctrl-c> stop”. Postprocess the results, and magnify the displacements 10x.

4. What is wrong with the contact interface? _____

Activate the flag IGAPF=2 on optional card “C” in the *CONTACT_... keyword. Repeat the simulation.

5. Does the job run? Why? _____

To reduce the severity of the initial contact, decrease the contact penalty scale factor using SLSFAC=0.01 on *CONTROL_CONTACT. Rerun the simulation.

6. Does the interface separate cleanly? _____

Plot the resultant interface force for the slave side of the interface. Save the curve data.

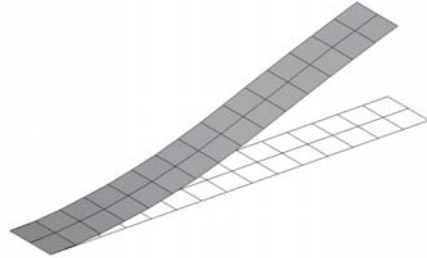
7. Why is the curve not smooth? _____

Restore the convergence tolerance DCTOL to it’s default value of 0.001, and repeat the simulation. Compare the new interface force with the saved data.

8. Is the new curve smoother? Why? _____

LS-DYNA Implicit Workshop

Problem #7: Implicit – Explicit Switching



Objective

- Learn to use the implicit method for static initialization of an explicit simulation.
- Learn to switch formulations during a simulation.
- Learn to perform a simple eigenvalue analysis.

Problem Description

A cantilevered strip of shell elements is loaded using the static implicit method. The analysis type is then switched to explicit, the load is removed, and the dynamic response is simulated. The first fundamental response frequency is verified by eigenvalue analysis.

Input Filename: ie_shell.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. How is the implicit – explicit switching activated _____
2. When does the simulation run implicit? Explicit? _____
3. How many implicit steps will be taken? _____
4. How is load applied? When is load removed? _____

Run the simulation, and postprocess the results. Plot the Y-displacement of a tip node vs. time.

5. Estimate the period of response for the first fundamental mode: _____ seconds.

Modify the input deck to conduct an eigenvalue analysis. Solve for the lowest five eigenvalues. Repeat the simulation. Observe the names of the two new output files which are created for eigenvalue analyses.

6. Which two new output files are created? _____

View the file eigout using a text editor.

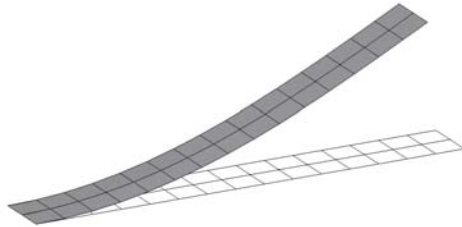
7. What is the period of the first fundamental mode? _____

Postprocess the binary file d3eigv using LS-POST. View the mode shapes.

8. What is the meaning of the “time” associated with each mode shape? _____

LS-DYNA Implicit Workshop

Problem #8: Springback using DYNAIN file



Objective

- Learn to generate a DYNAIN output file at the end of a simulation.
- Learn to apply artificial stabilization in a multi-step springback simulation.

Problem Description

This exercise involves two simulations. First, a cantilevered strip of shell elements is loaded using a dynamic explicit simulation. An output file named “dynain” is created at the end of this simulation. A second, implicit simulation is then performed which reads the dynain file and computes the springback deformation.

Input Filename: cant_dynain_load.k, cant_dynain_springback.k

Procedure

Copy the input files to your local directory. Using an editor, view the first input file and answer the following questions:

1. What keyword is used to create the DYNAIN file? _____
2. What material model is used? _____

Run the first simulation, and postprocess the results.

3. What is the initial Y-coordinate for all nodes? _____
4. What is the maximum effective stress at the end of the loading? _____

View the contents of the DYNAIN file which was created at the end of the first run.

5. Which keywords are used in the DYNAIN file? _____

View the contents of the second input file, and answer the following:

6. Could this file be easily created from the first input file? How? _____
7. How many steps will be used in this simulation? _____
8. How is load applied in this simulation? _____

Run the second input file, making sure the DYNAIN file produced by the first run is available for *INCLUDEing, and postprocess the results.

9. Is the termination time reached? Why? _____

Activate automatic time step control and artificial stabilization, and repeat the simulation. Postprocess the results, and plot the Y-coordinate of a tip node vs. time.

10. Does the springback deflection occur uniformly, or abruptly? _____
11. After springback: max. effective stress = _____ max. Y-coordinate = _____

Modify the artificial stabilization scale factor to SCALE = 0.050, and repeat the springback simulation. Postprocess, and plot the Y-coordinate of the tip node again.

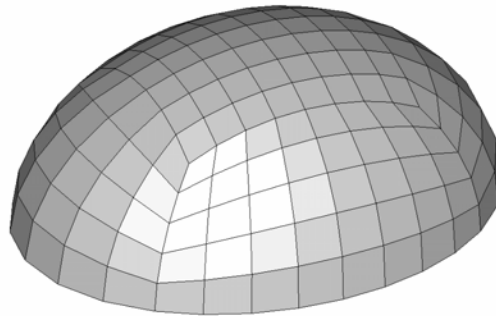
12. How has the evolution of springback deflection changed? _____
13. After springback: max. effective stress = _____ max. Y-coordinate = _____

Modify the nonlinear convergence test to compare the displacement increment to the total displacement over the current step (DNORM=1), repeat the simulation, and postprocess.

14. After springback: max. effective stress = _____ max. Y-coordinate = _____

LS-DYNA Implicit Workshop

Problem #9: Ellipsoidal Dome



Objectives

- Learn the behavior of different element formulations in static implicit simulations.
- Learn how to minimize hourglass problems.

Problem Description

A static load is applied to the center of an ellipsoidal dome. Shell elements are used. Nodes at the base of the dome are constrained, and included in a NODFOR output database.

Input Filename: ellipse.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. What is the maximum value of the applied load? _____
2. How many load steps are used? _____
3. What shell element formulation is used? _____

4. What hourglass control type is used? _____

Run the simulation, and postprocess the results (Note: NODFOR file contains reaction forces at constrained base nodes.)

5. What happens? _____

6. total reaction force _____ center displacement _____

Repeat the simulation using the S/R Hughes-Liu shell (ELFORM=6), and postprocess the results.

7. total reaction force _____ center displacement _____

Repeat the simulation using the Fast Fully Integrated shell (ELFORM=16), and postprocess the results.

8. total reaction force _____ center displacement _____

9. What conclusion can you draw about the effect of hourglass deformation on overall structural stiffness for this problem? _____

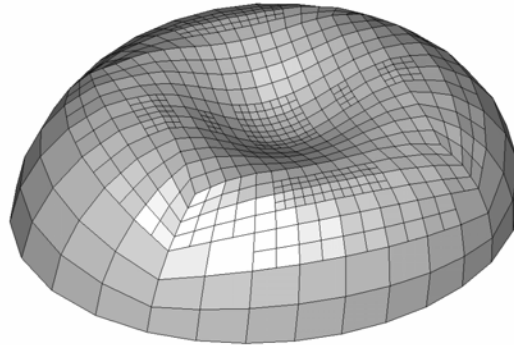
Concentrated applied loads are often responsible for initiating hourglass deformation. Edit the input deck and replace the single concentrated nodal load with pressure acting on the adjacent segments (Hint: Helpful keywords are included in the comments inside the input deck). Repeat the simulation using the default shell.

10. What happens? _____

11. total reaction force _____ center displacement _____

LS-DYNA Implicit Workshop

Problem #10: Adaptive Ellipsoidal Dome



Objectives

- Learn to activate mesh adaptivity in an implicit simulation.
- Learn how to minimize hourglass problems.

Problem Description

A static load is applied to the center of an ellipsoidal dome. Shell elements are used. Nodes at the base of the dome are constrained, and included in a NODFOR output database. Adaptivity is used to automatically refine the mesh in areas of high curvature.

Input Filename: aellipse.k

Procedure

Copy the input file to your local directory. Using an editor, view the input file and answer the following questions:

1. How frequently will the mesh be evaluated for refinement? _____
2. How many times can each element be subdivided? _____
3. How do you indicate which parts will be adapted? _____

4. Which element formulation is used? _____

5. How is load applied? _____

Execute the simulation, and view the results with the postprocessor.

6. How many time steps and cycles were used? _____ steps _____ cycles

7. Applied load _____ Center displacement _____

8. Does the adaptive mesh improve the hourglassing problem _____

Switch to pressure driven load application, and repeat the simulation. Postprocess the results. Using the NODFOR database, verify that the load is applied correctly as the mesh is refined.

9. How many time steps and cycles were used? _____ steps _____ cycles

10. Does the pressure load improve nonlinear convergence? _____

11. Applied load _____ Center displacement _____

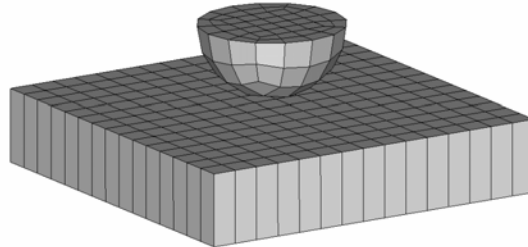
Experiment with shell element formulation #16.

12. Does shell type #16 improve hourglassing? _____

13. Does shell type #16 improve convergence behavior (number of steps/cycles)? _____

LS-DYNA Implicit Workshop

Problem #11: Ball Pressed Into Plate



Objectives

- Learn when implicit methods can run faster than explicit methods.
- Learn to troubleshoot contact and convergence problems.

Problem Description

A hemispherical ball of brick elements is displaced into a plate of brick elements. The plate is supported around its edges.

Input Filename: hemi.k

Procedure

Copy the input file to your local directory. The simulation is initially set up to run explicitly. Start the job and estimate how long it will take to finish. Stop the job using “<ctrl-c> stop” when your patience runs thin.

1. How long will the explicit job take to run? _____

Edit the input deck and activate the implicit method. Run the simulation and postprocess the results.

2. Does the problem run to completion? _____

3. What is the meaning of the “negative eigenvalue” warning message? _____

Modify the input deck to perform an eigenvalue analysis. Extract the lowest 30 modes.

4. Are any “zero” or “rigid body” modes detected? _____

5. What was the problem with this model? _____

6. Why did this model run as an explicit analysis? _____

Use *BOUNDARY_SPC_SET to add appropriate constraints to the ball (use the existing node set ID #1.) Repeat the simulation.

7. Examine the contact condition (HINT: cut a section plane normal to the Y-direction using SPLANE buttons). How deeply does the ball penetrate the plate? _____

To reduce penetration, use the penalty scale factor to increase the contact interface penalty stiffness by 10x (SLSFAC=1.0 on *CONTROL_CONTACT). Repeat the simulation.

8. How long does the simulation take to run using implicit method? _____

9. Is the penetration problem improved? _____