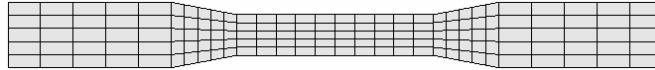


# LS-DYNA Implicit Workshop

## *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? default (Belytschko-Tsay type 2)
2. How many steps are used to apply the load? one

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 1000.0 Tip displacement 2.23e-2

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

4. Applied load 100,000 Tip displacement 2.27
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 (X) geometric  
( ) contact

Modify the \*CONTROL\_IMPLICIT\_SOLUTION keyword to perform a linear simulation, and record the results below for the large applied load:

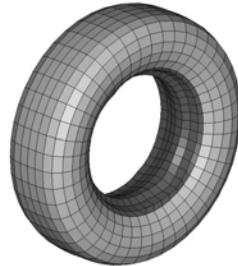
7. What is the reaction force at the fixed end of the beam? 98,940
8. Why does the reaction force not equal the applied load? linear analysis with nonlinear element (cross-sectional area has changed)

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

9. Does the reaction force now match the applied load? (X) 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? SPC constraints at bottom, pressure spike on lower sidewall

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

<b>method</b>	<b>2a. termination time</b>	<b>2b. CPU time</b>
EXPLICIT	<u>0.10</u> seconds	<u>505</u> 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? 8.63 Hz      What is it's period? 0.116 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	<u>1.0</u> seconds	<u>79</u> 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? Yes, until explicit TERM=0.10  
 6. How much faster is the implicit simulation for this (tiny) model? 505/7.9 = 64x  
 7. How could results of the eigenvalue analysis be used to select the timestep and output interval? Need at least 10 steps per period for shortest period of interest

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? Both, in this case

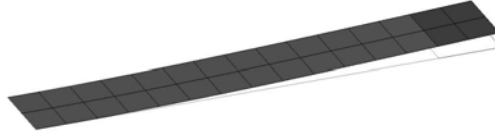
*Optional:*

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	<u>0.10</u> seconds	<u>300</u> seconds
same, but ISTIF=9999	<u>1.0</u> seconds	<u>163</u> 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? \*CONSTRAINED\_NODAL\_RIGID\_BODY
2. How long is the beam? 5 inches

Run the simulation using the original input file.

3. What type of analysis was performed? Linear static
4. What is the tip deflection? 0.1635 inches

5. What is the angle of rotation of the rigid body (Note: rotations in the RBDOUT file are given in radians)? 4.8445e-2 rad = 2.776 degrees
6. What is the maximum X-stress at the supported end? 22,463 psi
7. What is the maximum X-stress in the “rigid” elements? -38,677 psi

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? 0.163 in
9. What is the angle of rotation of the rigid body? 2.769 degrees
10. What is the maximum X-stress at the supported end? 22,424 psi
11. What is the maximum X-stress in the “rigid” elements? 7.0e-9 psi

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? 0.164 in
13. What is the maximum X-stress at the supported end? 22,463 psi
14. What is the maximum X-stress in the tip elements? 775 psi

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.)

(see rigid\_tip\_solution.ppt)



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

5. What is happening at the time shown above? plastic yielding through X-section

6. max end displacement 7.07 max eff. stress 317 max eff. strain 9.62%

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? “<ctrl-c> nlprint” or NLPRINT=1

8. How many cycles are used in the simulation? 2366

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 7.0 max eff. stress 310 max eff. strain 8.61%

10. Why is this problem easier to solve? prescribing displacements eliminates unknowns

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? Normal termination, 23 steps, 413 cycles

12. max end displacement 7.06 max eff. stress 317 max eff. strain 9.61%

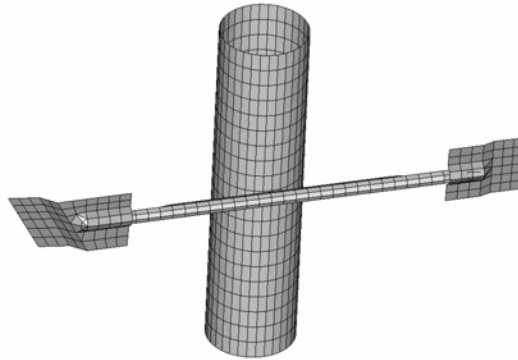
Using the “ASCII” menu, load the GLSTAT database and plot the step size vs. time.

13. When does the step size change? Why? Quickly rises to DTMAX=0.05 because less than 200 iterations are needed for each step.



# 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? **NODES\_TO\_SURFACE because doorbeam normals point everywhere**
2. Why is the doorbeam chosen as the slave side? **It's normals are most erratic**

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? Sticky (400 vs 656 cycles)

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? Simply retract the indenter, since analysis is static
5. At what time is the full load applied? (HINT: Check the input file.) 100
6. Why are the maximum reaction force and springback predictions from this simulation misleading? Didn't compute a solution at peak load

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”.

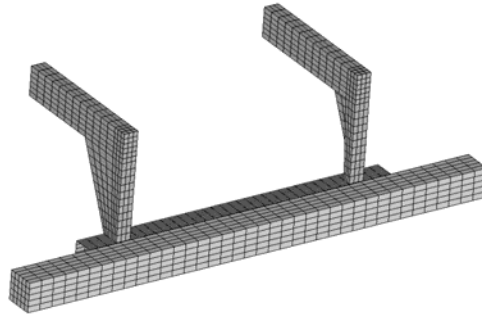
7. Is the peak load computed at the correct time? yes
8. Why is the force-deflection behavior not smooth? Poor equilibrium estimate

Select the alternate displacement tolerance scheme using DNORM=1 on \*CONTROL\_IMPLICIT\_SOLUTION, and repeat the simulation.

9. Does the alternative displacement tolerance become (X) 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? More strict tolerance (d3hsp: ||u||=1.e-4 @t=1.0, compared with ||u||=1.e+0 in 7 above.)
11. What is the springback deflection at the center of the doorbeam? 3.38

# LS-DYNA Implicit Workshop

## *Problem #6: Truck Bumper*



---

### *Objectives*

- Learn the behavior 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? \*CONTACT\_SURFACE\_TO\_SURFACE
2. Is the thickness of the shell elements considered? Yes (SHLTHK=1)

Run the simulation and postprocess the results.

3. Why does the simulation stop? Convergence failure at step #1

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? Sticking along lower edge

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

5. Does the job run? Why? No, energy explosion at first contact

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? yes

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

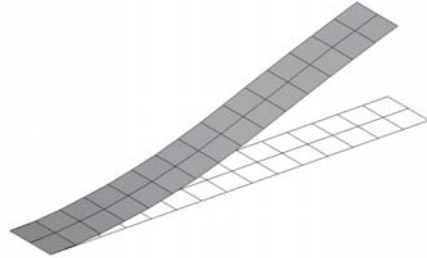
7. Why is the curve not smooth? Poor estimate of equilibrium (sloppy tolerance)

Restore the convergence tolerance DCTOL to its 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? Yes, more accurate equilibrium

# 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 manual method, curve ID=9
2. When does the simulation run implicit? Explicit? Implicit until t=0.01, then explicit
3. How many implicit steps will be taken? 20
4. How is load applied? When is load removed? End force, max @ t=0.01, then zero

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: 0.0075 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? d3eigv, eigout

View the file eigout using a text editor.

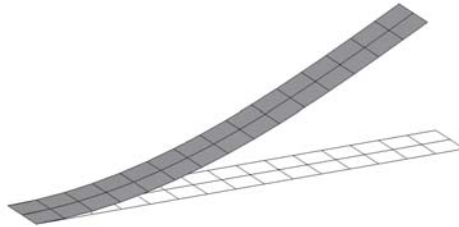
7. What is the period of the first fundamental mode? 0.00744 sec

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? Freq (cycles)

# 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 creates the DYNAIN file?  
\*INTERFACE SPRINGBACK LSDYNA
2. What material model is used? Elastic

Run the first simulation, and postprocess the results.

3. What is the initial Y-coordinate for all nodes? 0.0
4. What is the maximum effective stress at the end of the loading? 146,475

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? \*NODE, \*ELEMENT\_SHELL\_THICKNESS, \*INITIAL\_STRESS\_SHELL

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? Subset of initial input, plus \*CONTROL\_IMPLICIT section
7. How many steps will be used in this simulation? 4
8. How is load applied in this simulation? Internal initial stress

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? No, convergence failure step 1, 80 iter.

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? Abrupt at end of sim.
11. After springback: max. effective stress = 2081 max. Y-coordinate = 6.67e-3

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? More gradual
13. After springback: max. effective stress = 5765 max. Y-coordinate = 7.3e-2

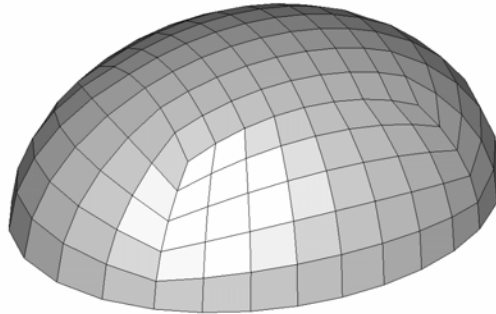
Modify the nonlinear convergence test to compare the displacement increment to the total displacement over the current step (DNORM=1), repeat the simulation, and postprocess.

14. After springback: max. effective stress = 1997 max. Y-coordinate = 7.43e-4



# 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? 500,000
2. How many load steps are used? 100 + auto
3. What shell element formulation is used? default (B/T)

4. What hourglass control type is used? type 4 (stiffness)

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

5. What happens? Normal termination, 19 steps, 280 cycles

6. total reaction force 499,951 center displacement -11.78

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

7. total reaction force 499,936 center displacement -11.37

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

8. total reaction force 500,005 center displacement -10.85

9. What conclusion can you draw about the effect of hourglass deformation on overall structural stiffness for this problem? Hourglassing produces "softer" response

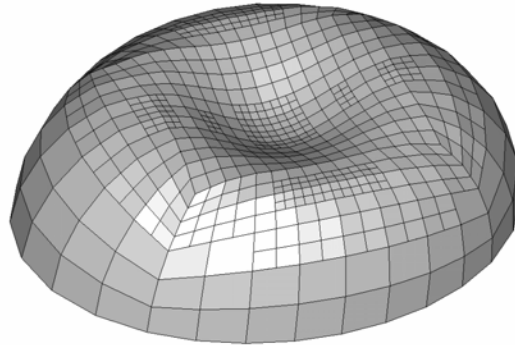
Concentrated applied loads are often responsible for initiating hourglass deformation. Edit the input deck and replace the single concentrated nodal load with pressure acting on the adjacent segments (Hint: Helpful keywords are included in the comments inside the input deck). Repeat the simulation using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

10. What happens? Normal termination, 15 steps, 247 cycles

11. total reaction force 519,437 center displacement -10.20

# 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? 0.10 sec
2. How many times can each element be subdivided? 2
3. How do you indicate which parts will be adapted? ADPOPT=1 on \*PART

4. Which element formulation is used? type 2 (B/T)

5. How is load applied? Point load

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

6. How many time steps and cycles were used? 16 steps 203 cycles

7. Applied load 519,089 Center displacement -11.29

8. Does the adaptive mesh improve the hourglassing problem better, still bad locally

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? 17 steps 372 cycles

10. Does the pressure load improve nonlinear convergence? no

11. Applied load 519,089 Center displacement -11.29

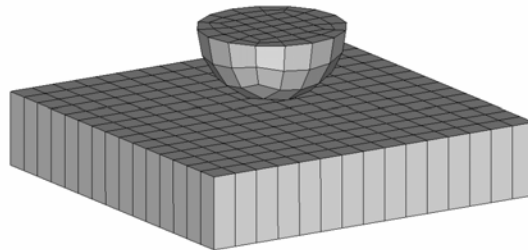
Experiment with shell element formulation #16.

12. Does shell type #16 improve hourglassing? Hourglassing not bad with pressure load

13. Does shell type #16 improve convergence behavior (number of steps/cycles)? Yes (14 steps, 149 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? About 8 minutes

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

2. Does the problem run to completion? No, negative eigenvalue error message, then divergence

3. What is the meaning of the “negative eigenvalue” warning message? Singular stiffness matrix

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

4. Are any “zero” or “rigid body” modes detected? Yes, three
5. What was the problem with this model? Forgot to add constraints dx=dy=rz=0
6. Why did this model run as an explicit analysis? Rigid body modes OK for dynamic

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? 0.29

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? 4 cpu seconds
9. Is the penetration problem improved? Yes, g=-0.061