# Quasi-Static Analysis of Can Crushing Using Ansys Workbench LS-DYNA: Step-by-Step Tutorial

Hesam Moghaddam

Developed and curated by the Ansys (part of Synopsys) Academic Program

education@ansys.com

## Ansys Software Used

This resource uses Ansys LS-DYNA® nonlinear dynamics structural simulation software and Ansys Mechanical™ structural finite element analysis software.

## Summary

This example presents a detailed analysis of the quasi-static crushing of a thin-walled aluminum can by a rigid punch. It has two parallel objectives. First, it introduces the physical concepts behind quasi-static analysis and explains why slow loading is essential when inertia effects corrupt the response. Second, it walks the reader through the complete workflow for setting up and solving the problem in Ansys Workbench using the LS-DYNA solver. The study begins by defining the quasi-static assumption and the verification criteria based on energy balance and force equilibrium. It then describes the geometry and materials for the soda-can crushing problem and explains the differences between dynamic (impact) and quasi-static loading. Detailed sections describe how to prepare the model in the Ansys Workbench tool: importing geometry, assigning materials and contacts, applying boundary conditions, and defining the time-dependent velocity profile. The core of the document compares several analyses with different strain rates (progressively longer end times of 0.0006 s, 0.06 s, 0.6 s and 1.2 s) while maintaining the same overall punch displacement. For each case the response is analyzed by plotting contact forces, kinetic and internal energies, and the ratio of kinetic to internal energy. The final sections discuss best practices for explicit quasi-static analysis and summarize the key findings.

## Table of Contents

## 1. Introduction

In many structural simulations, the goal is to understand the load-carrying capacity of a component under slowly applied loads. When the process occurs gradually—such as in metal forming, bending, or crushing—dynamic inertia effects are often negligible, and the response can be considered quasi-static. In practice, deformation processes with characteristic strain rates on the order of $10^{-6}$ to $10^{-3}$ $s^{-1}$ are typically classified as quasi-static, whereas processes involving much higher strain rates ($\geq 10^1$ $s^{-1}$) exhibit significant inertial effects and must be treated as dynamic.. In an ideal quasi-static analysis the external work goes into deforming the structure rather than into accelerating masses. The explicit dynamics solver in Ansys LS-DYNA software can be used for such problems, provided that the loading is applied slowly enough and that certain verification checks are performed. If the loading is too rapid, the simulation will generate artificial oscillations and inflated reaction forces; the results will reflect an impact event rather than a slowly applied load. In a dynamic finite element analysis, a structure's motion obeys the matrix equation of motion:

$$\mathbf{M}\ddot{\mathbf{u}}\left(\mathrm{t}\right)+\mathbf{C}\dot{\mathbf{u}}\left(\mathrm{t}\right)+\mathbf{K}\mathbf{u}\left(\mathrm{t}\right)=\mathbf{P}\left(\mathrm{t}\right)$$

where **M**, **C**, and **K** are the assembled mass, damping and stiffness matrices; **u**, **u̇**, and **ü** are vectors of nodal displacements, velocities and accelerations, respectively; and **P** is the vector of external nodal loads. In explicit solvers this second-order ordinary differential equation is integrated through many small increments using the central-difference scheme. To start a new increment, the explicit algorithm computes the nodal accelerations from the equilibrium of internal and external forces at the current time:

$$\mathbf{a}_{\mathrm{n}}=\mathbf{M}^{-1}\left(\mathbf{P}_{\mathrm{n}}-\mathbf{F}_{\mathrm{n}}^{\mathrm{int}}\right)$$

where $\mathrm{F}_{\mathrm{n}}^{\mathrm{int}}$is the internal force vector assembled from the elements. Because the mass matrix in explicit codes such as the Ansys LS-DYNA solver is lumped (diagonal), its inverse is trivial to compute. Once $a_n$ is known, the nodal velocities and displacements are advanced in time by half-step updates:

$$\mathbf{v}_{\mathrm{n}+\frac{1}{2}}=\mathbf{v}_{\mathrm{n}-\frac{1}{2}}+\mathbf{a}_{\mathrm{n}}\Delta\mathrm{t},\ \ \mathbf{u}_{\mathrm{n}+1}=\mathbf{u}_{\mathrm{n}}+\mathbf{v}_{\mathrm{n}+\frac{1}{2}}\Delta\mathrm{t}$$

where Δt is the time increment. These formulas show why the method is called "explicit": the new kinematic state depends only on known quantities from the previous increment. The stability of the central-difference operator is conditional—the time step must remain smaller than a critical value inversely related to the highest natural frequency of the mesh; in practice the estimate $\Delta t \leq L_c/c$ (the transit time of a dilatational wave across the smallest element dimension $L_c$) is often used. This is called Courant–Friedrichs–Lewy condition.

### 1.1 Quasi-static approximation

When the loading changes very slowly, inertial and damping effects can be neglected; the governing equation reduces to the static equilibrium relation

$$\mathbf{K\,u=P}$$

This is the basis of quasi-static analysis: the load history is stretched sufficiently in time that the kinetic energy remains small, and the dynamic solution approaches the static response. In explicit codes the full dynamic equation is still integrated in time, but the prescribed load or displacement history is ramped smoothly over a long duration—often using a half-sine function—and the ramp time is chosen to be at least 1.5–2 times the fundamental period of the structure. Under such conditions, the inertial term $\mathbf{M}\ddot{\mathbf{u}}$ and the damping term $\mathbf{C}\dot{\mathbf{u}}$ contribute negligibly to the overall equilibrium, and the solution converges with the static response. Mass scaling can be used to increase the stable time step, but any artificial mass must be introduced judiciously and its effect on the solution checked.

## 1.2 Practical implications

In practice, quasi-static simulations using an explicit solver still follow the central- difference- integration algorithm described above. The difference lies in the loading profile and the verification of quasi-static conditions. Slow, prescribed motions minimize kinetic energy and allow the explicit formulation to converge toward a static solution, while the conditional stability of the time integration dictates how small $\Delta t$ must be. By monitoring the ratio of kinetic energy to internal energy and ensuring it remains small, and by checking that reaction forces balance the applied loads, engineers can confirm that the explicit simulation is behaving in a quasi-static manner.

To assess whether a simulation is quasi-static, two simple measures are commonly used:

1. **Energy ratio.** During a quasi-static process the kinetic energy (KE) should remain much smaller than the internal (strain) energy (IE). A rule of thumb is that the ratio KE/IE should stay below 2 % for most of the simulation. Ansys LS-DYNA software provides tools to plot internal, kinetic and total energy versus time and to check this ratio. If kinetic energy becomes comparable to the internal energy, inertia is influencing the solution, and the load must be applied more slowly.
2. **Force equilibrium.** Newton's third law requires that the sum of all applied forces equals the sum of reaction forces at supports. In an explicit dynamic simulation inertia forces appear when accelerations are present; these inertia forces can cause discrepancies between applied and reaction forces. A quasi-static analysis should exhibit close agreement between the reaction and applied force histories in all directions, with only small fluctuations due to numerical noise. Plotting the punch–can contact force and the can–die reaction force together reveals whether action and reaction balance.

These verification criteria guide the remainder of this case study. In the sections that follow we will see how high-speed loading violates the criteria and how progressively slower loadings lead to quasi-static behavior.

# 2. Problem Description

The system of interest in this case study consists of three bodies: a thin-walled aluminum can, a rigid punch, and a rigid die. The can is modeled as a shell with uniform thickness (0.25 mm in this study) and an outer diameter of 66 mm and height of 115 mm—dimensions typical of a beverage can. The material is an aluminum alloy with elastic–plastic behavior approximated using the piecewise linear plasticity model in Ansys LS-DYNA software. The punch and die are modeled as rigid bodies using solid elements made of structural steel. Both are unyielding during the analysis and serve to impart deformation into the can. Figure 1 illustrates the assembled geometry of the punch–can–die system. The can sits on the die, and the punch approaches from above to crush it. Frictional contact occurs between the punch and can and between the can and die. A coefficient of friction of 0.3 is used for both interfaces. Rigid-body motion of the can is prevented by constraining the die in all directions. The punch is controlled via a prescribed velocity history; no force or pressure load is applied directly.

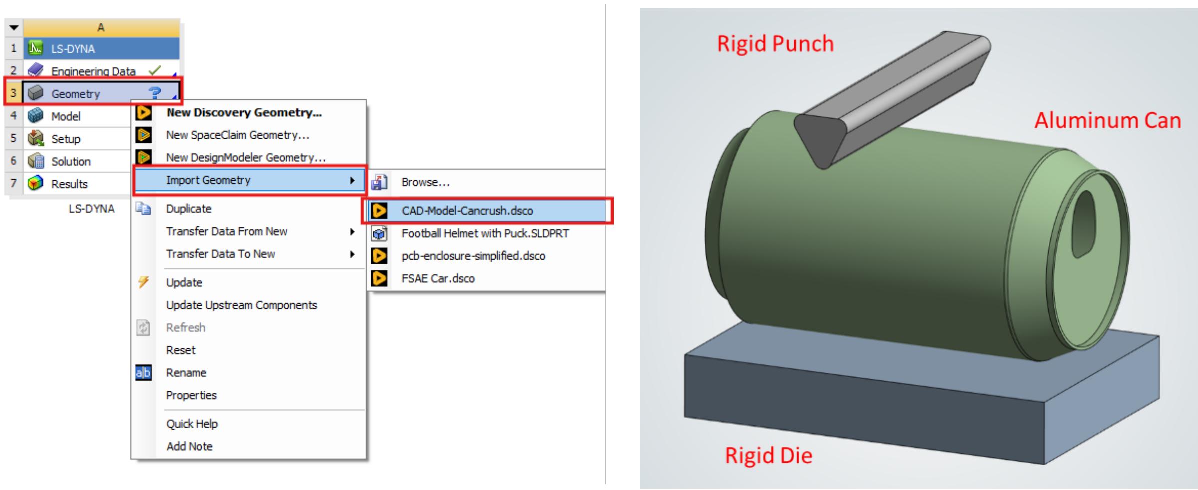
## 3. Simulation Setup in Ansys Workbench LS-DYNA Solver

The Ansys Workbench environment provides a convenient framework for setting up explicit dynamic simulations using the LS-DYNA solver. In this section, the complete model setup for the can-crushing problem is described. Rather than presenting a strict click-by-click procedure, the workflow is explained in a tutorial-style narrative to clarify both what is done and why each step is required.

Begin by launching Ansys Workbench 2025 R2 (or a later version). From the Toolbox, drag an LS-DYNA analysis system into the Project Schematic.

### 3.1 Importing Geometry

The CAD file "CAD-Model-Cancrush.dsco" is provided with the project files. To import the geometry, right-click on the Geometry cell in the Project Schematic, as shown in Figure 1(a), and select Import Geometry. Once the import is complete, the model should appear as shown in Figure 1(b).
The geometry consists of three parts: a rigid punch, a thin-walled aluminum can, and a rigid die, as illustrated in Figure 1. The can is positioned between the punch and the die, with the punch aligned vertically above the can.




(a) (b)

Figure 1. Geometry of the can-crushing problem. (a) Importing geometry in Ansys Workbench module; (b) assembled CAD model consisting of a rigid punch, aluminum can, and rigid die.

### 3.2 Material Assignment

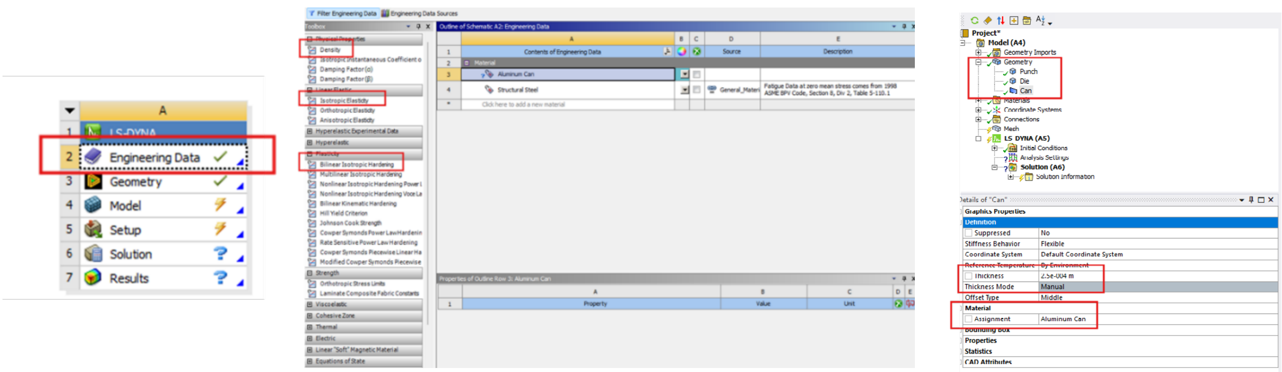
Two materials are used in this analysis. Additional materials can be added later if the model is extended or modified. Material definitions are created under the Engineering Data cell in the Project Schematic, as shown in Figure 2(a). The can material is defined as an aluminum alloy with elastic–plastic behavior, modeled using Isotropic Elasticity combined with Bilinear Isotropic Hardening. The punch and die are assigned Structural Steel and treated as rigid bodies. After defining the materials, open the Model cell to launch Ansys Mechanical software. Select each part under the Geometry branch and assign the appropriate material in the Details panel. For the punch and die, set Stiffness Behavior = Rigid to ensure that they remain undeformed during the simulation. The material properties used in this study are summarized in Table 1.

Table 1. Material properties for the can, punch, and die.

| Material | Density (kg/m³) | Young's modulus, E (GPa) | Poisson's ratio, ν | Yield strength (MPa) | Tangent Modulus (MPa) |
|---|---|---|---|---|---|
| Structural Steel | 7850 | 200 | 0.3 | N/A | N/A |
| Aluminum alloy | 2710 | 70 | 0.3 | 290 | 0 |

## 3.3 Opening the Model for Preprocessing and Postprocessing

When the model is first opened in Ansys Mechanical software, a question mark, "?", appears next to the can geometry in the Outline, indicating missing information. This is because the can is modeled as a shell and requires thickness definition. Also, make sure to check for the unit system on the top toolbar and make sure it's the right one. Select the can geometry and enter a shell thickness of 0.25 mm in the Details panel, as shown in Figure 2(b). At this stage, it is also good practice to rename the geometry parts to Punch, Can, and Die for clarity. Make change the behavior for die and punch to "Rigid" under Stiffness Behavior.




(a) (b)

Figure 2 Material definition in Ansys Workbench interface;
(b) assigning materials and shell thickness in Ansys Mechanical software

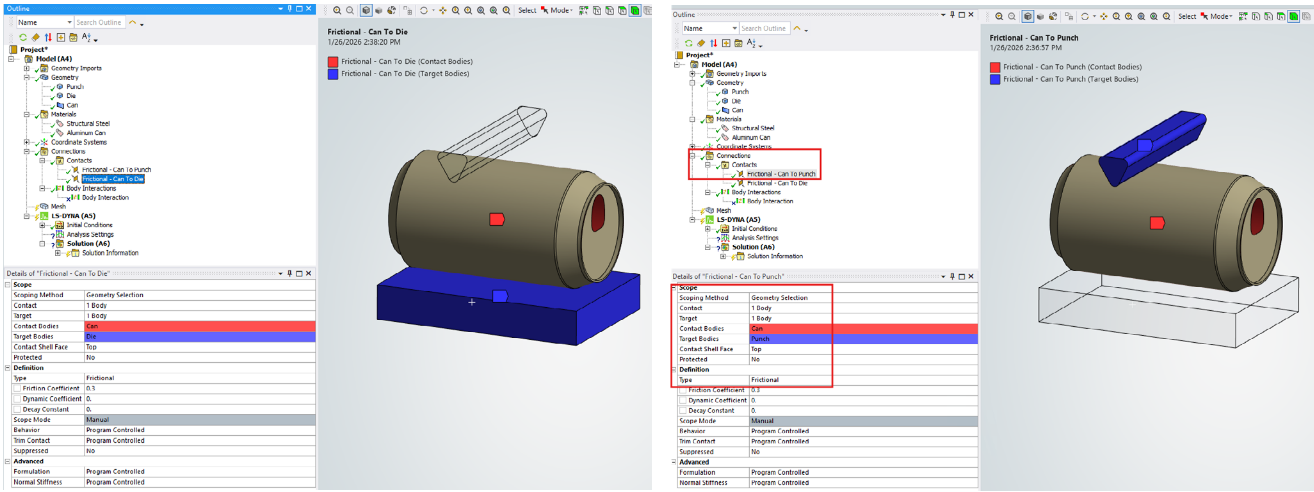
## 3.4 Contacts

Contact interactions can be defined using either Body Interaction objects or explicit Contact Regions. By default, Ansys LS-DYNA software automatically creates a Body Interaction, which should first be suppressed to avoid duplicate contact definitions.

For this problem, surface-to-surface frictional contact is defined between:

- the punch and the can, and
- the can and the die.

To create these contacts, right-click on Connections in the Outline and select Insert → Manual Contact Region. Change the default contact type from Bonded to Frictional and set the coefficient of friction to 0.3. Assign the can as the Contact surface and the punch as the Target surface. Rename the contact based on its definition, then duplicate it and modify the second contact to represent the can–die interface. For both contacts, set Contact Shell Face = Top. This means that for a shell body, ANSYS will use the top surface of the shell as the contact surface in the contact pair. The final contact definitions are shown in Figure 3. Only one contact formulation should be active at a time; mixing Body Interaction and Contact Regions can lead to unintended behavior.

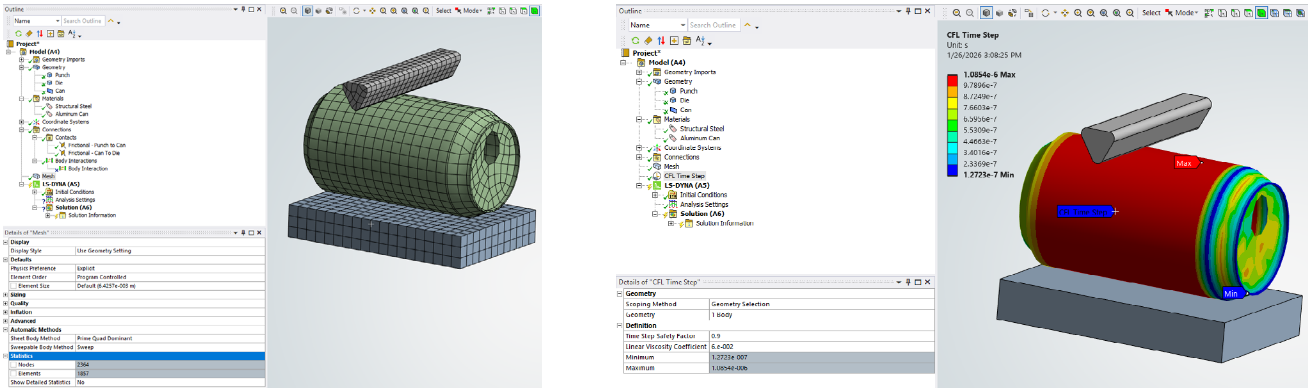



(a) (b)

Figure 3. (a) Punch-to-Can contact; (b) Can-to-Die contact

## 3.5 Meshing

Since we are using Ansys LS-DYNA solver which solves using explicit methods, mesh size plays a critical role in explicit analysis because the stable time step is directly related to the smallest element dimension. For this study, the default automatic mesh in Ansys Mechanical software provides sufficient mesh quality. Ansys Mechanical software might outputs a message in the message center that “Some of the elements on the problematic bodies can't meet the specified target metrics. Please check the elements and try changing the mesh size settings to achieve the needed mesh quality.” means that the mesher could not achieve the requested element quality criteria (skewness, aspect ratio, orthogonal quality, etc.) on some regions of your model. This warning can generally be ignored if only a very small percentage of elements (e.g., less than 1%) fail the target quality metrics, they are not located in critical regions such as contact areas, high-stress zones, supports, or load application points, the solver converges normally, and the results remain stable after performing a mesh refinement (mesh independence) check. Right-click on Mesh and select Update. The resulting mesh consists primarily of quadrilateral shell elements for the can and solid elements for the rigid bodies, as shown in Figure 4(a). A mesh sensitivity study can be performed later if higher accuracy is required. To estimate the stable time step, insert a CFL Time Step under the Model branch and scope it to the can only. The minimum and maximum allowable time steps are shown in Figure 4(b). CFL data is not relevant for rigid bodies and therefore is not evaluated for the punch and die.




(a) (b)

Figure 4. default mesh generated in Ansys Mechanical software; (b) CFL time-step estimate for the can.

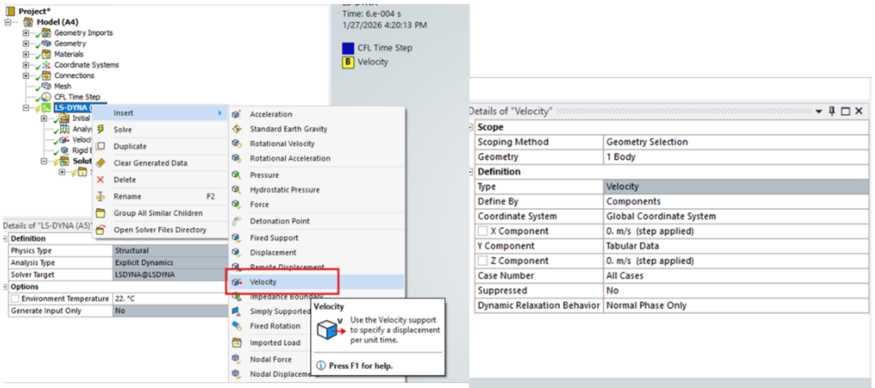
## 3.6 Boundary Conditions and Loading

Boundary conditions and loading are defined under the LS-DYNA branch in the Outline. The Analysis Settings initially display a warning "?" symbol, indicating that required inputs are missing. Open this item and set the End Time to 6e-4 s (0.6 ms) for the fast-loading case. All other settings are left at their default values, with no mass scaling or hourglass control applied at this stage.

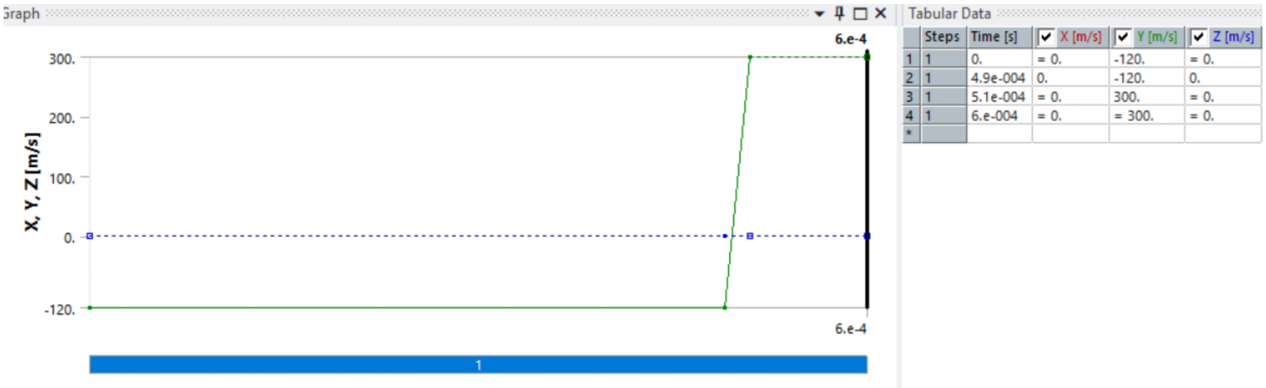
Rigid-body motion of the system is constrained by fixing the die. Insert a Rigid Body Constraint under LS-DYNA → Insert → Rigid Body Tools, scope it to the die, and constrain all translational and rotational degrees of freedom. The Can inherits this constraint through contact with the die.

The punch motion is controlled using a prescribed velocity rather than a force or displacement. This approach provides direct control over the loading rate, which is essential for quasi-static studies. Using LS-DYNA → Insert →Velocity , define the boundary condition, scope it to the punch, and activate only the Y-direction. Make sure that X and Z components are set to "Constant" value of 0 and then Y is set to tabular. This will prompt the tabular data where you can enter the step velocity profile by entering the time steps and corresponding velocity values.

Figure 5(b) shows the velocity history used for the fast-loading (impact) case. The punch velocity ramps from approximately -120 m/s to 300 m/s over the duration of the simulation. When designing a quasi-static analysis the velocity should ramp up smoothly during loading and unloading without sudden jumps, i.e. that the loading velocity is constant, and its magnitude should be chosen so that the kinetic energy remains small compared to the internal energy. The next sections demonstrate how this profile is modified for slower simulations.




(a)




(b)

Figure 5. (a) Defining a tabular velocity in Ansys Mechanical software; (b) velocity profile for the fast-loading case (end time = 0.0006 s).

## 4. Solution

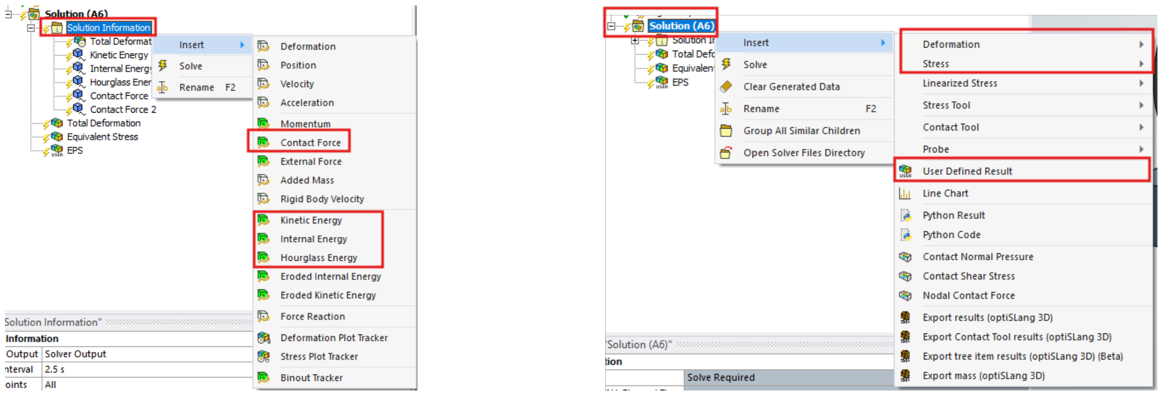
To post-process the results, several output quantities are requested before solving. Under the Solution branch, insert:

- Total Deformation,
- Equivalent (von Mises) Stress, and
- Equivalent Plastic Strain defined using a User-Defined Result with the expression of EPS types in the highlighted box.

These results can be scoped to either all bodies or only the can, depending on the desired output. Figure 6(a) illustrates the setup of these contour plots.

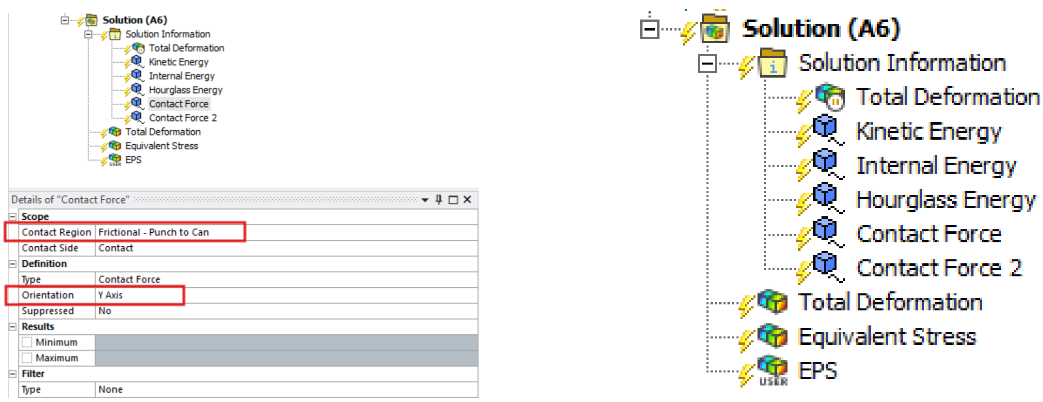
In addition, verification of quasi-static behavior requires monitoring energies and contact forces. Under Solution Information, enable Kinetic Energy, Internal Energy, Hourglass Energy, and Contact Force. Scope all energy monitors to the can. For contact force, create two plots: one for the punch–can contact and one for the can–die contact under "Contact Region", both oriented in the Y-direction, set under "Orientation", as shown in Figure 7(a).

Once all results are defined, the model is ready to solve. Right-click on Solution and select Solve, or use the Solve button from the toolbar.




(a) (b)

Figure 6. (a) Setting up the contour plot results for post-processing
(b) Setting up the energy and contact force plots (time histories).




(a) (b)

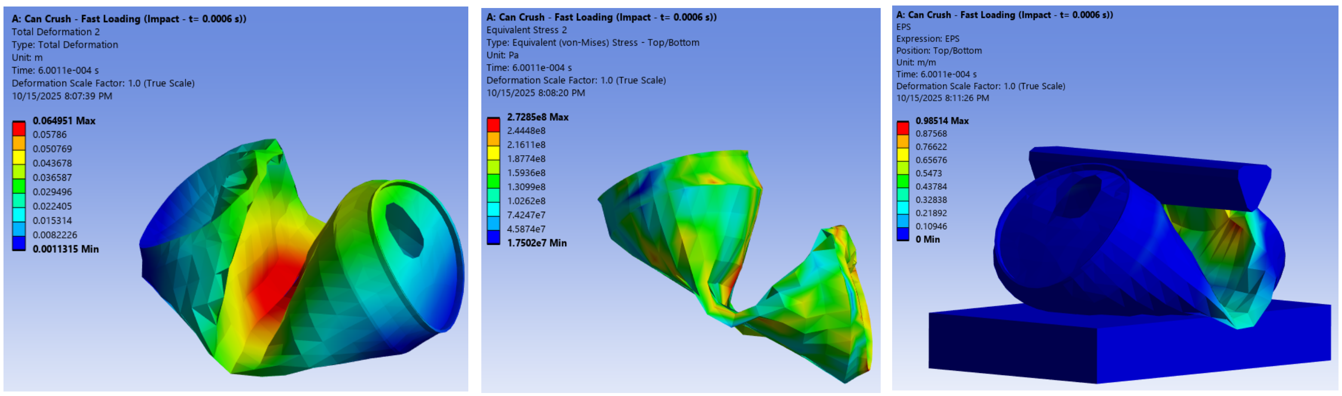
Figure 7. (a) Contact force monitor setup;
(b) overview of Solution and Solution Information items before solving.

## 5. Postprocessing: Fast Loading (Impact) Analysis

The initial analysis replicates an impact scenario. The total time of loading and unloading is set to 0.0006 s, and the velocity profile in Figure 5(b) drives the punch to achieve the target 59 mm displacement during the loading time of 0.00049 seconds, followed by unloading at the end time of 0.00006 seconds. Such rapid loading causes the punch to strike the can at approximately 120 m/s. Though this case may be interesting from an impact perspective, it does not satisfy the quasi-static criteria.

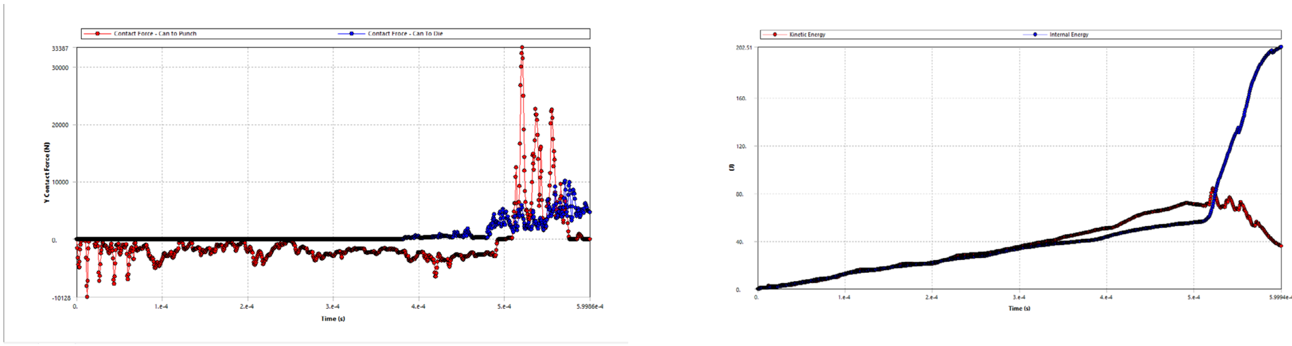
Figure 8 shows some sample results for the impact case. Figure 8(a), (b), and (c) show contour plots of total deformation, von Mises equivalent stress and effective plastic strain at the end of the 0.0006 s simulation (Fast Loading/ Impact Scenario). The deformed shape shows the folding of a thin-walled can under axial crushing. The regions under the punch and against the die exhibit high compressive stress and plastic strains, while the sidewalls undergo complex bending and buckling.

After running the fast-loading simulation, examine the Contact Force history. Hold control and select both Contact Force plots to see what is shown Figure 9(a). This plots the force transmitted between the punch and the can and between the can and the die. The action–reaction forces should coincide if the motion is quasi-static. In this case the force histories exhibit large oscillations and overshoots. The reaction force (i.e. can-die contact force) lags the applied force (i.e. punch-die contact force) because of inertia; therefore, Newton's third law is not satisfied at each instant. These oscillations are evidence of wave propagation and dynamic ringing inside the can. You can also use the Chart option to plot these. Navigate to the top toolbar and under "Solution" tab, select Chart. When prompted, Choose the Objects, aka results, that you want to plot. In the "Outline Selection". Then you can adjust Chart Controls, Axis Labels, and other options as shown in Figure 9(c). The Kinetic versus Internal Energy is plotted using the chart option in Figure 9(d).Next, we inspect the energy time histories. In the fast-loading case the kinetic energy is of the same order of magnitude as the internal energy for much of the simulation. This violates the rule of thumb that the kinetic energy should remain below 2–5 % of the internal energy. When internal and kinetic energy oscillate together, the solution is dominated by dynamic effects. Because inertia dominates, the deformed shape of the can (not shown here) exhibits significant elastic rebound and unrealistic buckling patterns compared to a slowly crushed can. The conclusion is clear: the fast impact analysis is unsuitable for estimating quasi-static crushing resistance.




(a) (b) (c)

Figure 8. Contour plots of (a) Total deformation (m); (b) von-Mises stress distribution (Pa); and ( c) Equivalent plastic strain (m/m). High stress and strain concentrations occur under the punch and at the folds, while the overall deformation pattern exhibits axisymmetric folding. The can forms a series of axisymmetric folds with high plastic strains.




(a) (b)

Figure 9. Time history plots of (a) Contact force histories for the fast impact case. Large oscillations and mismatches between the punch–can force and the can–die reaction force indicate that inertia effects dominate; (b) Kinetic Energy to Initial energy ratio

## 6. Time Scaling and the Quasi-Static Assumption

To obtain a quasi-static solution using an explicit solver, we must slow down the loading so that inertia effects diminish. However, we still want the punch to achieve the same total displacement of about 59 mm. In explicit dynamics the maximum stable time step is limited by the smallest element dimension and the speed of sound in the material (the Courant–Friedrichs–Lewy condition). By prescribing slower motion, we allow the system to respond gradually and reduce kinetic energy. This strategy is sometimes called time scaling.

Time scaling involves adjusting both the end time of the simulation and the velocity profile so that the total displacement remains constant. For example, if the fast impact case uses an end time of 0.0006 s and a peak velocity of 120 m/s, a simulation with an end time of 0.06 s (100 times longer) should use a peak velocity of approximately 1.2 m/s to reach the same final displacement. The velocity profile should also ramp up smoothly over a longer duration to avoid sudden accelerations. The same idea extends to an end time of 0.6 s (1,000 times longer) and 1.2 s (2,000 times longer).

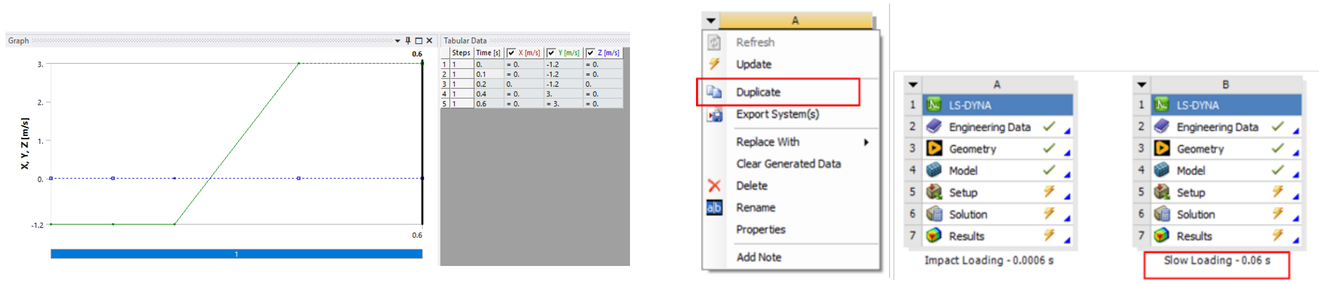
Although slower end times require more computational time, they are still feasible because explicit dynamics solvers can handle large numbers of time steps efficiently. More importantly, slowing the loading reduces kinetic energy and allows the solution to approach the quasi-static limit. The following sections show how the response evolves as the end time increases from 0.06 s to 1.2 s.

## 7. Slow Loading Scenarios

### 7.1 Intermediate Loading: End Time = 0.06 s

The first slow-loading scenario increases the end time from 0.0006 s to 0.06 s—a two-order-of-magnitude increase. The velocity profile is scaled down accordingly: its peak value is about 1.2 m/s, and the ramp is spread out over several milliseconds. All other settings (geometry, materials, contacts) remain unchanged. The velocity profile is shown in Figure 10 (a) and can be used to set up the simulation.

To preserve these settings for comparison, duplicate the analysis system before making any changes; in the project tree right-click the system and choose Duplicate. You can create multiple copies for the different end times that will be studied. Figure 10(b) shows the steps to Duplicate the system analysis in the Ansys Workbench tool. After duplicating the system analysis, rename it to “Slow Loading – 0.06 s”.




(a) (b)

Figure 10. (a) Velocity profile for the Slow Loading scenario;
(b) How to duplicate the system analysis in Ansys Workbench

Next, double click on the Model cell and open the Ansys Mechanical software. First, go to analysis setting, change the end time to 0.06. Then, go to “Velocity” and update the time and velocity values

based on the values shown in Figure 10 (a). Solve the problem and look at the results under Solution and analyze the energy time history and contact forces under Solution Information.
Figure 11 shows the contact force history for this intermediate case. Compared to the fast impact simulation, the oscillations are greatly reduced. The punch–can force and the can–die reaction force follow each other more closely, though there are still discrepancies near the peaks. Some inertia effects persist, reflecting the fact that the can is still being loaded relatively quickly.

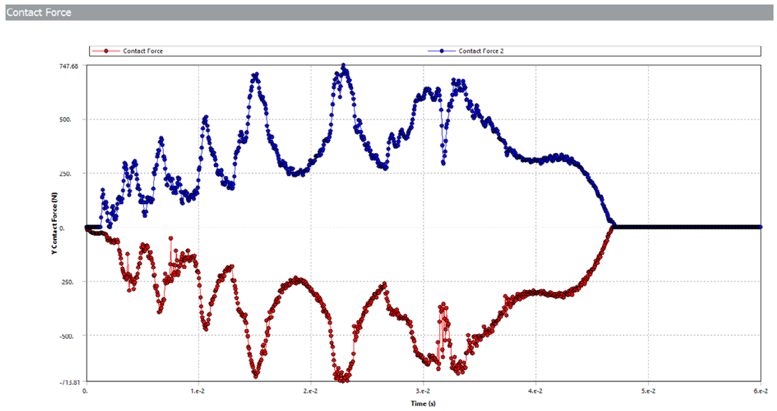
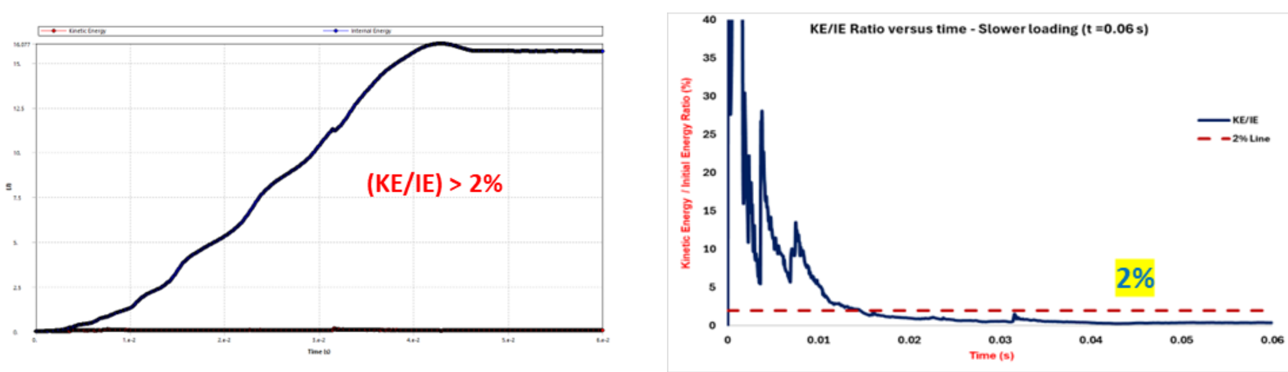



Figure 11. Contact force histories for the intermediate loading case (end time = 0.06 s). The oscillations are smaller than in the impact case, but the punch–can and can–die forces do not completely coincide.

The energy ratio plot, Figure (12a), provides further insight. For much of the simulation the kinetic energy is less than about 10 % of the internal energy. While this is a significant improvement over the impact case, it still exceeds the recommended 2% threshold during portions of the loading. This is more visible in Figure 12 (b) where it shows a rescaled and zoomed in view of the energy ratio. Consequently, the 0.06 s analysis is not yet fully quasi-static. Nonetheless, it illustrates that simply increasing the end time reduces oscillations and brings the solution closer to equilibrium. To Rescale the plot and obtain what is shown in Figure 12 (b), you need first to export the plot as an Excel file and then use Excel to rescale your plot. To accomplish this, Inside Ansys Mechanical software, and under "Solution Information", right click on any of the plots that you defined before, such as Contact Force, or Kinetic Energy, and click on Export as Text File and choose Excel File (*xls) under "Save as Type".



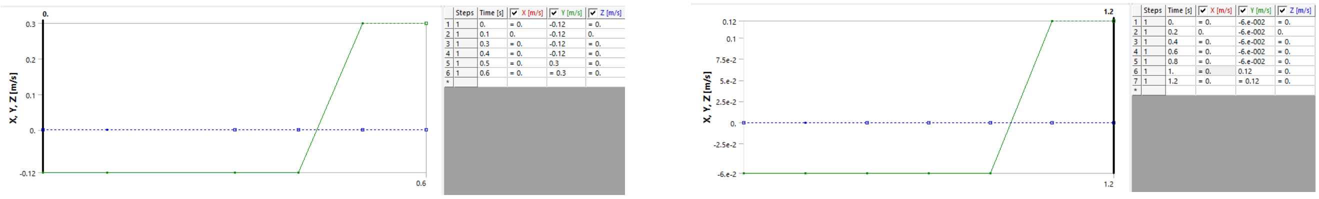

(a) (b)

Figure 12. (a) Kinetic energy to internal energy ratio for the 0.06 s simulation. (b) Zoomed in and Rescaled: the ratio is improved compared to the impact case but still not below 2% for most of the simulation to justify quasi-static behavior.

## 7.2 Slower Loading (end time = 0.6 s) and Slowest Loading (end time = 1.2 s)

To reduce inertia further, we will make the loading slower. W will consider a Slower Loading Scenario" which uses an end time of 0.6 s, an order of magnitude longer than the previous case. Finally, we will do the "Slowest Loading" scenario where the velocity profile is applied for 1.2 seconds, twice slower than the last one. The velocity profile is scaled down, with a peak velocity around 0.3 m/s and 0.12 m/s, respectively. The punch travels the same total distance but at a much slower rate.

Figure 13 (a) and (b) shows the velocity profile for the Slower and Slowest loading scenarios, respectively. You can follow the same steps as described in previous section to duplicate the system analysis and update the velocity profile in Ansys Mechanical software to solve these two scenarios.



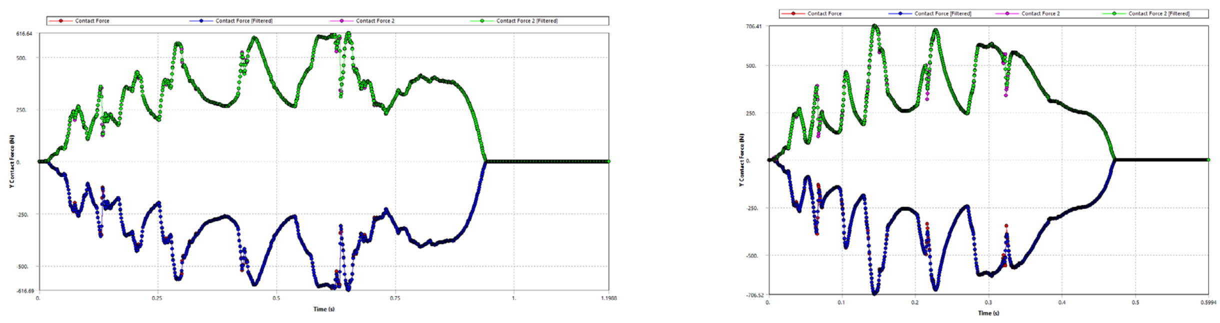

| | Steps | Time [s] | X [m/s] | Y [m/s] | Z [m/s] |
|---|---|---|---|---|---|
| 1 | 1 | 0. | = 0. | -0.12 | = 0. |
| 2 | 1 | 0.1 | 0. | -0.12 | 0. |
| 3 | 1 | 0.3 | = 0. | -0.12 | = 0. |
| 4 | 1 | 0.4 | = 0. | -0.12 | = 0. |
| 5 | 1 | 0.5 | = 0. | 0.3 | = 0. |
| 6 | 1 | 0.6 | = 0. | = 0.3 | = 0. |
| * | | | | | |

| | Steps | Time [s] | X [m/s] | Y [m/s] | Z [m/s] |
|---|---|---|---|---|---|
| 1 | 1 | 0. | = 0. | -6.e-002 | = 0. |
| 2 | 1 | 0.2 | 0. | -6.e-002 | 0. |
| 3 | 1 | 0.4 | = 0. | -6.e-002 | = 0. |
| 4 | 1 | 0.6 | = 0. | -6.e-002 | = 0. |
| 5 | 1 | 0.8 | = 0. | -6.e-002 | = 0. |
| 6 | 1 | 1. | = 0. | 0.12 | = 0. |
| 7 | 1 | 1.2 | = 0. | = 0.12 | = 0. |
| * | | | | | |

(a) (b)

Figure 13. (a) Velocity profile for (a) the Slower Loading scenario; (b) Slowest Loading scenario

Figure 14 compares the contact force equilibrium plot for the Slower and Slowest scenarios. Figure 14 (a) shows that the contact forces now almost coincide in both scenarios with the Slowest Loading (t = 1.2 s) showing the most equilibrium and perfectly balanced. Oscillations are minimal and the difference between the punch–can force and the can–die reaction force is small. This indicates that inertia forces no longer dominate the responses.  For completeness, a very slow loading case with an end time of 1.2 s is considered. The peak velocity is reduced to around 1.2 m/s. This case serves as a verification that the results obtained at Slower Loading scenario (t = 0.6 s) are independent of the loading rate.



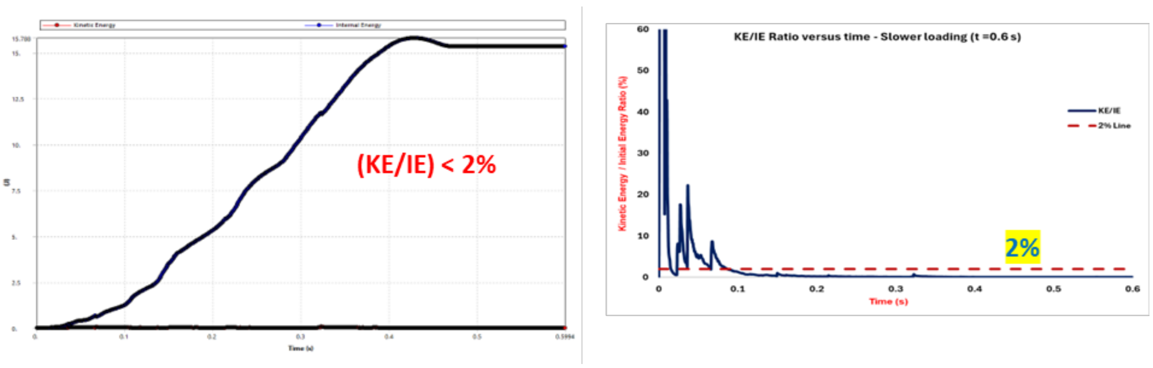

(a) (b)

Figure 14. Contact force equilibrium plots for the (a) Slower and (b) Slowest scenarios.

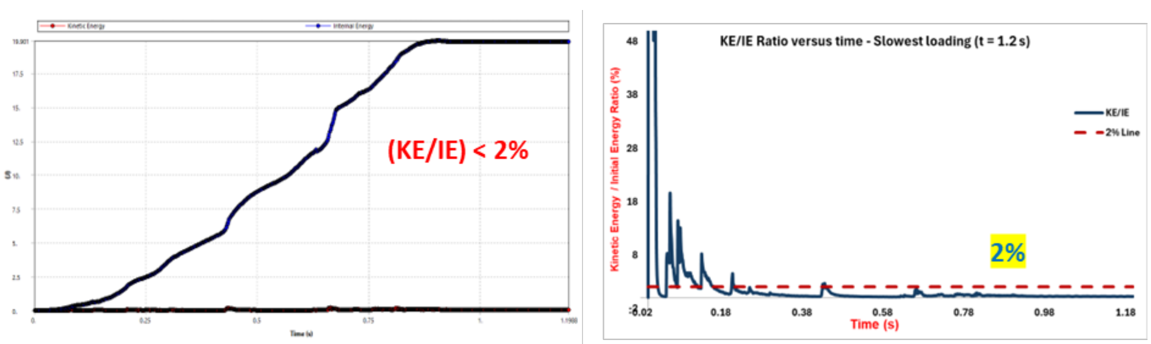
The 0.6 s case therefore represents a practical compromise: it yields quasi-static behavior while keeping the computational cost reasonable. Only minor residual oscillations remain near the end of the punch stroke, which can be further suppressed with in Ansys Mechanical software. In explicit dynamics simulations, even when the loading is intended to be quasi-static, the numerical solution often contains high-frequency oscillations caused by elastic wave propagation, contact chatter, and the conditional stability of the central-difference time integration scheme. To aid interpretation of force and energy histories, Ansys Mechanical software provides a built-in Butterworth low-pass filter for time-history plots under Solution Information. The Butterworth filter is commonly used because it has a maximally flat frequency response in the passband, meaning it smooths high-frequency numerical noise without introducing artificial ripples or phase distortion in the lower-frequency response that governs the quasi-static behavior.

Click on any of the plots under “Solution Information” and under Details, choose Butterworth under Filter and set the cut-off frequency to 20,000 Hz. The Butterworth low-pass filter is used to suppress high-frequency numerical noise caused by stress-wave propagation and contact oscillations that are inherent to explicit time integration. A cut-off frequency of 20,000 Hz is sufficiently high to preserve the low-frequency structural response governing quasi-static behavior, while effectively removing non-physical, high-frequency content. The filter is applied only during post-processing and does not alter the underlying solution; it is used solely to improve the clarity of force and energy time-history plots once quasi-static conditions have been verified from the unfiltered results.

Figure 15 shows that the kinetic-to-internal energy ratio remains below 2 % throughout the simulation; the kinetic energy is almost negligible. There is essentially no difference between the results of the 1.2 s and 0.6 s simulations. This confirms that the 0.6 s analysis is sufficiently slow to capture the quasi-static response.




(a)




(b)

Figure 15. Kinetic-to-internal energy ratio for the (a) Slower (t = 0.6 s) and (b) Slowest (t = 1.2 s) simulation. The ratio remains below about 2 % which is within the recommended range for quasi-static range. The right pictures show the zoomed-in and rescaled version of the plot on the left.

## 8. Summary of Time-Scaling Studies

The sequence of simulations reveals a clear trend: as the end time (and thus the duration of the loading) increases, the kinetic energy decreases relative to the internal energy and the contact forces approach equilibrium. Compared to the fast loading scenario(i.e. impact), with t = 0.0006 s, The 0.06 s case reduces dynamic effects but does not fully eliminate them; the 0.6 s and 1.2 s cases both satisfy the quasi-static criteria. Because the 1.2 s simulation yields no significant difference from the 0.6 s case, the shorter 0.6 s simulation is deemed adequate for engineering purposes.

Readers implementing their own quasi-static analyses should adopt a similar strategy: start with an initial guess for the end time, run the simulation, and examine energy ratios and force equilibrium. If the kinetic energy is too high or the reaction and applied forces do not match, increase the end time and adjust the velocity profile accordingly. Iteratively slowing the load until the results converge is the most reliable approach to achieve quasi-static behavior.

## 9. Best Practices for Explicit Quasi-Static Analysis

The can-crushing case study illustrates several general principles that apply to quasi-static analysis with explicit solvers:

- **Prescribe motion rather than force.** Applying a displacement or velocity boundary condition gives direct control over the loading rate and avoids the instability that can arise from force-controlled loading. In this tutorial the punch motion is prescribed via a velocity table; the solver computes the reaction force.
- **Use smooth loading curves.** Ramp the velocity gradually at the beginning and end of the loading to avoid sudden accelerations. Sharp changes create high-frequency content that increases kinetic energy.
- **Avoid contact closure at high velocity.** If the initial gap between contacting bodies closes too quickly,an impact will occur and dynamic effects will dominate. Adjust the starting position or slow the ramp so that contacts are engaged gently.
- **Keep frictional contacts consistent.** When using Body Interaction or Contact definitions, suppress unused contact definitions to prevent duplicate contact calculations. Choose an appropriate coefficient of friction; in this study 0.3 was used for both interfaces.
- **Monitor energies and forces.** Always plot internal and kinetic energies and the ratio KE/I. Plot reaction and applied forces to check equilibrium. If necessary, use numerical filters in Ansys Mechanical software such as the Butterworth Filter as explained to remove residual oscillations from the curve.
- **Perform time-scaling studies.** Quasi-static behavior cannot be assumed a priori; it must be verified by comparing simulations with increasing end times. Increase the end time until the results converge and the kinetic energy stays below the threshold. The final proof of quasi-static behavior is a run with an even longer end time that yields the same results.

Adhering to these practices ensures that explicit quasi-static analyses produce reliable results without unnecessary computational expense. The can-crushing example demonstrates how a simple change in end time (from 0.06 s to 0.6 s) can transform a dynamic simulation into a quasi-static one.

## 10. Conclusions

This tutorial presented a comprehensive case study of quasi-static can crushing using the LS-DYNA solver in Ansys Workbench. Starting from an impact-like analysis with an end time of 0.0006 s, the study demonstrated how dynamic inertia effects manifest through large oscillations in contact forces and high kinetic energies. By systematically increasing the end time and scaling the velocity profile, the analysis transitioned from a dynamic to a quasi-static regime. At an end time of 0.6 s and beyond, the kinetic energy remained below 3 % of the internal energy, and the reaction and applied forces balanced almost perfectly. The final deformation pattern exhibited realistic folding and plastic strain distributions typical of a crushed can.

The key lessons are:

- Quasi-static analyses require careful control of loading rates. **Time scaling** is an effective technique for reducing kinetic energy without altering the displacement outcome.

Verification is essential. Plotting **energy ratios** and **force equilibrium** provides objective measures of quasi-static behavior. A simulation cannot be considered quasi-static unless the kinetic energy remains small and action and reaction forces match.
Once quasi-static behavior is verified, the detailed stress and strain fields can be trusted for design analysis and optimization.

By following the procedures outlined in this tutorial, educators and students can confidently use Ansys LS-DYNA software in explicit mode to investigate slow crushing and forming problems. The approach emphasizes understanding and verification rather than rote button clicks, aligning with the educational philosophy of the Ansys Educator Hub.

**Use and Reproduction**



**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