Summary
The purpose of this example is to compare different studies with flexible or rigid bodies. The method for using the flexible bodies in an explicit analysis is also studied.

At first, the truck is modeled using a classical finite element model for explicit analysis. All parts of the truck are modeled using different kinds of finite elements, such as shells, bricks, springs and beams. The volumes monitored with perfect gas characterize the tires.

The problem is divided into two loading phases. First, gravity is applied as a quasi-static load. Then, the truck’s Virtual Proving Ground (VPG) is studied to observe the truck driving over an obstacle (bump).

For the gravity loading phase, the explicit approach using relaxation techniques or not is employed. For the VPG analysis, three approaches are compared: (i) classical finite element model; (ii) simplified finite element model with a global rigid body; and (iii) finite element model involving a flexible body. The last approach requires the first run to compute the Eigen and static modes. A flexible body input file is then generated for use in a second time-history run. The main interest of this method is to economize the CPU time.

The following studies are depicted in this tutorial:

- VPG with a complete finite element model
- VPG with flexible and rigid bodies
VPG with a Complete Finite Element Model

**Title**
VPG with a complete finite element model

**Number**
14.1

**Brief Description**
After applying gravity, a truck runs on a horizontal plane and passes over a bump.

**Keywords**
- Shell, brick, beam, beam type spring, monitored volume (perfect gas)
- Quasi-static load treatment, kinetic relaxation
- Type 7 and 2 interfaces, auto-impacting, rigid wall (infinite plane and cylinder)
- Linear elastic law (/MAT/LAW1), elasto-plastic law (/MAT/LAW2), void material law (/MAT/LAW0)

**RADIOSS Options**
- Boundary conditions (/BCS)
- Gravity (/GRAV)
- Initial velocity (/INIVEL)
- Kinetic relaxation (/KEREL)
- Monitored volume type gas perfect (/MONVOL/GAS)
- Rigid body (/RBODY)
- Rigid wall (/RWALL)
- Skew frame (/SKEW)

**Input File**
VPG_complete_model:
<install_directory>/demos/hwsolvers/radioss/14_Truck_with_FXB/VPG_complete_model/TRUCK*

**RADIOSS Version**
51j

**Technical / Theoretical Level**
Advanced

**Overview**
Physical Problem Description
In the first step, the truck model is placed on the ground under the gravity field until static equilibrium is obtained. Then, under the impulse of 15.6 m/s (56 km/h) initial speed, the truck runs in a straight line and passes over a speed bump. The shock is expected to cause major deformation in some highly solicited parts.

Units: mm, s, ton, N, MPa.

In order to simplify modeling, most of the parts undergo the linear elastic material law (/MAT/LAW1).

- Young’s modulus: 205000 MPa
- Poisson’s ratio: 0.3
- Density: 7.85x10⁹ Kg/l

The elasto-plastic Johnson-Cook model (/MAT/LAW2) mainly describes the joint and strengthening elements, such as the beams and spring.

- Young’s modulus: 205000 MPa
- Poisson’s ratio: 0.3
- Density: 7.85x10⁹ Kg/l
- Yield stress: 180 MPa
- Hardening parameter: 480 MPa
- Hardening exponent: 0.5

The truck represents a simplified model having the essential parts. The weight of the truck is approximately 8 tons.
Analysis, Assumptions and Modeling Description

Modeling Methodology

Finite Element mesh:
The truck model is meshed with 21430 elements - 148356 degree of freedom, as follows:

- 1D elements: 173
- 2D elements: 20109
- 3D elements: 1148
Details of the elements used are provided in Table 1 below:

Table 1: Composition of the EF mesh.

<table>
<thead>
<tr>
<th>Element</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node</td>
<td>24726</td>
</tr>
<tr>
<td>4-node shell</td>
<td>18471</td>
</tr>
<tr>
<td>3-node shell</td>
<td>1638</td>
</tr>
<tr>
<td>Brick</td>
<td>1148</td>
</tr>
<tr>
<td>Beam</td>
<td>47</td>
</tr>
<tr>
<td>Spring</td>
<td>126</td>
</tr>
<tr>
<td>Part</td>
<td>159</td>
</tr>
</tbody>
</table>

The improved Belytschko hourglass formulation (type 4 hourglass, $I_{\text{nat}}=4$) is used for shell elements in the explicit computation. The Eigen analysis requires fully-integrated elements since the computation mode needs an implicit option. Compatible element formulations are set by default.

Fig 3: Overall mesh of truck.

The main parts of the model are shown in the table below:

<table>
<thead>
<tr>
<th>Part</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Trailer body</td>
<td><img src="image" alt="Trailer body" /></td>
</tr>
<tr>
<td>Cab</td>
<td><img src="image" alt="Cab" /></td>
</tr>
</tbody>
</table>
Monitored Volumes / Perfect Gas

Monitored volumes are used to model the pressure in the tires. They are defined with one or more shell property sets and the surface must be closed. The monitored volume used is the perfect gas type.

The main properties for this type are:

- External pressure: 0.1 MPa
- Initial internal pressure: 0.3 MPa
- Constant gas: 1.4

All other properties are set to default values. The parts modeled with the monitored volumes are highlighted in Fig 5:
Connections Between Parts

In order to assemble the parts, four link types are used in the model:

- Beam type spring (type 13)
- Rigid body (kinematic condition)
- Tied interface type 2 (kinematic condition)
- Merged nodes

The beam type spring elements are useful for modeling the welding points. The modeling techniques are described in the RADIOSS User's Guide.

In this example the beam type spring properties are:

- Young’s modulus: 210000 MPa
- Inertia: $2 \times 10^{-4}$ kg.m$^2$
- Mass: $2 \times 10^{-6}$ ton

Force and moment are read from the input curves:

Table 2: Input force versus displacement curve.

<table>
<thead>
<tr>
<th>Displacement</th>
<th>-1</th>
<th>0</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx, Fy, Fz</td>
<td>$-10^6$</td>
<td>0</td>
<td>$10^6$</td>
</tr>
</tbody>
</table>

Table 3: Input moment versus rotation curve.

<table>
<thead>
<tr>
<th>Rotation</th>
<th>-1</th>
<th>0</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mx, My, Mz</td>
<td>$-10^6$</td>
<td>0</td>
<td>$10^6$</td>
</tr>
</tbody>
</table>
Fig 6: Beam type springs (13) used in the model.

The type 2 tied interface rigidly connects a set of slave nodes to a master surface. The kinematic constraint is set on the slave nodes which remain in the same position on their master segments. This interface is a kinematic condition. The Spotmag spotweld formulation is set to zero in order to connect two meshes without coincident nodes. The master surface should be the coarser mesh.

Fig 7: Tied interface (type 2).

Fig 8: Tied interfaces (type 2) used in model
Rigid bodies are created to join two or more parts together. For these rigid bodies no added mass is required and the master node can be located anywhere. Slave nodes may not accept the other kinematic conditions (such as tied interface).

A spherical inertia must be used for the rigid bodies having only two slave nodes for ensuring the stability of the connected elements ($I_{sphe} = 1$). Thus, inertia is spherical and not computed from data.

**Contact Modeling – Auto-impacting**

Taking into account self-impacting parts, a type 7 auto-impacting interface must be used. The Block Format definition of this interface is to define master surface (/SURF/PART), then define slave nodes as all nodes on this surface (/GRNOD/SURF).

Gap is equal to 0.5 mm.
Wheel Rotation Modeling
Wheels are linked to a frame using an axle attached to the brake systems. A beam element (in red in the opposite figure) models the axle causing released rotations at the node linked to the wheel rim.
RADIOSS Options Used

Two types of rigid walls are set up:

- A fixed infinite plane (ground);
- A fixed infinite cylinder having a diameter, D = 1500 mm (bump).

![Fig 13: Infinite plane and cylindrical wall for modeling the ground and bump (slave nodes displayed in green).](image)

The cylindrical wall is defined by point M (500, 0, -600), M1 (500, 100, -600) and the diameter.

![Fig 14: Cylindrical wall definition.](image)

Both rigid walls are tied to allow the wheels to turn. The tire parts define the slave nodes for the infinite plane (contact of ground and tires) and only the nodes of the front right tire are set as slave for the speed bump in order to model a local bump. The obstacle is not infinite.

A kinematic condition is applied on each impacted slave node. Therefore, a slave node cannot have another kinematic condition; unless such condition is applied in an orthogonal direction. In such a manner, incompatible kinematic conditions can be detected, due to the coincident normal orientations along the Z axis of the cylindrical and plane walls. However, the common slave nodes are not affected simultaneously by both kinematic conditions.

![Fig 15: Incompatible kinematic conditions (no orthogonal directions of normals).](image)

A 15600 mm.s⁻¹ (56 km/h) initial velocity (/INIVEL) is applied to all nodes of the structure in the X direction at t = 0.3 s. This initial condition is defined in the D02 restart file (start time: 0.3 s), which is run after achieving the quasi-static equilibrium with gravity loading.
Option in \textbf{D02} file:

\begin{itemize}
  \item /INIV/TRA/X/1: initial translational velocities in the X direction
  \item 15600 of 15600 mm/s
  \item 1 265130 on node 1 to 265130 (/INIV/TRA/X/1)
\end{itemize}

Fig 16: Selected nodes for the initial translational velocity of the truck (56 km/h) at $t = 0.3$ s.

\textbf{Quasi-static Loading: Gravity Effect on Initial Static Equilibrium}

The quasi-static solution of gravity loading on structure deformation is the steady state part of the dynamic response and describes the pre-loading case before the transient analysis. Thus, simulation is divided into two phases: quasi-static response (structure deformation under the gravity effect) and dynamic behavior (run and impact on the bump). The solution is obtained using the kinetic relaxation method.

Gravity is applied to all nodes of the model. A constant function defines the gravity acceleration in the Z direction versus time and is equal to $-9810$ mms$^2$. Gravity is activated with the /GRAV option.

The explicit time integration scheme assumes starting with nodal acceleration computation. It is very efficient for simulating dynamic loadings. Nevertheless, quasi-static simulations via a dynamic resolution method need to minimize the dynamic effects in order to converge towards static equilibrium. Among the usual methods employed, the kinetic relaxation method is quite effective and is activated in the \textbf{D01} Engine file using /KEREL. All velocities are set to zero each time the kinetic energy reaches a maximum value.
**Simulation Results and Conclusions**

Animation of the passing over the speed bump:

*Fig 17: Kinetic relaxation method using the /KEREL option.*

*Fig 18: Distribution of von Mises stress on the model during bump passage.*
Fig 19: Cab deformation (initial state and after bump passage).
VPG with Flexible and Rigid Bodies

**Title**
VPG with flexible and rigid bodies

**Number**
14.2

**Brief Description**
After applying gravity, a truck runs on a horizontal plane and passes over a bump. The major part of the truck is described using a flexible body.

**Keywords**
- Eigen and static analysis
- Eigen modes
- Flexible body

**RADIOSS Options**
- Eigen modes computation (/EIG)
- Flexible body input file (/FXINP)
- Flexible body (/FXBODY)
- Rigid body (/RBODY)

**Input File**

VPG_Rigid_body:
<install_directory>/demos/hwsolvers/radioss/14_Truck_with_FXB/VPG_Rigid_body/TRUCK*

VPG_Flexible_body:
<install_directory>/demos/hwsolvers/radioss/14_Truck_with_FXB/VPG_Flexible_body/Model_EIG/TRUCK_EIG_*
<install_directory>/demos/hwsolvers/radioss/14_Truck_with_FXB/VPG_Flexible_body/Model_FXB/TRUCK_FXB_*

**RADIOSS Version**
51j

**Technical / Theoretical Level**
Advanced

**Overview**

---

15
Aim of the Problem

The purpose of this example is to perform an Eigen analysis on a complete truck model with the purpose of creating a flexible body which will be used to model the truck’s main part, excluding transmission (wheels, left-springs, differential, shaft, brakes and axles). In order to appreciate the quality of the modeling, the results will be compared with those obtained using two other models: one without a flexible body (previous analysis) and the other substituting the flexible body with a rigid body.

The study deals with:

- an Eigen analysis to create a file containing the dynamic response of the structure
- a quasi-static analysis (explicit pre-loading by gravity)
- an explicit dynamic analysis with a global flexible body
- an explicit dynamic analysis with a global rigid body

Analysis, Assumptions and Modeling Description

Modeling Methodology

The original model and two alternative models are compared:

| 1 complete model | 1 model including a global flexible body | 1 model including a global rigid body |

In the previous section where a complete finite element model is used, it is noted that the stress and strain levels are low for most parts of the global model. Thus, the CPU time can be considerably reduced if the elements working in the linear elastic field are replaced with a flexible body. The purpose of this example is to provide an overall view of using flexible bodies in RADIOSS.

The top part of the truck, where no damage and no plastic strain occurs, is first successively modeled with a rigid body (non-deformable) and then with a flexible body (deformable), as shown in Fig 20.

Parts of the truck covered by rigid or flexible body is shown in the following diagram:

![Diagram of truck model](image)

Red part = global flexible or rigid body

Fig 20: Top part of truck included in a flexible/rigid body depending on the model.

RADIOSS Options Used
Eigen and Static Modes Computation – Flexible Body Creation

A flexible body is similar to a rigid body where displacement is computed on nodes corresponding to vibration modes. The input file for a flexible body uses the RADIOSS Eigen modes and static modes computation. Modes can derive from experimental analysis, as well as from vibratory software.

The total displacement field for every point of a flexible body is obtained by displacing the local frame defining the rigid body modes and from an additional local displacement field corresponding to the body’s small vibrations.

A preliminary study with RADIOSS extracts Eigen or static modes for creating the flexible body input file used in a second run. This computation phase requires the /EIG and /FXINP options.

The /EIG option is set up in the Starter input file and defines the part to be included in the flexible body, as well as the type and number of modes to be computed.

In this example, the main data is:
- Number of modes = 25
- Maximum Eigen frequency = no
- Minimum Eigen frequency = 13 Hz
- Number of Eigen modes per block = 10

Two types of modes can be obtained:
- Free Eigen modes
- Static modes

Eigen modes (or dynamic modes) are computed for the entire structure without any specific boundary condition. The equation solved is:

\[ K \mathbf{u} = \omega^2 \mathbf{M} \mathbf{u} \]

In this approach, rigid body modes in the structure are possible and give null Eigen frequencies.
If \( Ku = 0 \), \( K \) is not singular and \( u_r \neq 0 \), therefore, \( \partial M u_r = 0 \) and \( \partial = 0 \)

In addition, static modes can be computed if boundary conditions are added to a node group in the flexible body frontier. They correspond to the static response of the structure. All degrees of freedom in the set of interface nodes concerned by the additional boundary conditions are fixed and one static mode is computed for each constrained degree of freedom. The equation solved is:

\[
K u = F
\]

Static modes are displayed with null frequencies in animations.

Rigid modes are not permitted and generate null pivots during inversion of the stiffness matrix.

It should be noted that modes computation requires the implicit options in the Engine file (\(/\text{IMPL/LINEAR}\) and \(/\text{IMPL/SOLVER/1}\)).

- Eigen frequencies are provided in the Engine listing file. One animation exists per computed mode.
  - The \(/\text{FXINP}\) option is used in the Engine file for creating a flexible body input file .fxb. The flexible body has the same support as that defined in /EIG. You should enter:
    - Identification number of the Eigen mode or static mode problem defined in /EIG; The critical structural damping coefficient used for computing the Rayleigh damping coefficient to be introduced in the flexible body (it is recommended to use default value 0.03);
    - Type of flexible body (1 = free flexible body, 2 = fixed flexible body). The flexible body input file can be used in a second run using \(/\text{FXBODY}\) in the Starter Input file to generate a flexible body. The flexible body input file name ending in .fxb for the RADIOSS format and master node coordinates are required (possible coordinates are given at the top of the .fxb file).

<table>
<thead>
<tr>
<th>Eigen Analysis (writing FXB input file)</th>
<th>Run using Flexible Body</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid bodies + master nodes.</td>
<td>Master nodes of rigid bodies.</td>
</tr>
<tr>
<td>Boundary conditions.</td>
<td>Master node of the flexible body.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>FXB domain can contain</th>
<th>FXB domain must not contain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free parts.</td>
<td>Rigid bodies (slave nodes).</td>
</tr>
<tr>
<td>Slave nodes on the flexible body frontier.</td>
<td>Slave nodes on the flexible body frontier.</td>
</tr>
<tr>
<td>Rigid body overlapping on flexible body and the rest of structure.</td>
<td>Rigid body overlapping on flexible body and the rest of structure.</td>
</tr>
<tr>
<td>Truss elements.</td>
<td></td>
</tr>
<tr>
<td>Void material.</td>
<td></td>
</tr>
<tr>
<td>Monitored volumes.</td>
<td></td>
</tr>
</tbody>
</table>

Options incompatible with the implicit solver must be avoided.
The inputs files used with the specific options are:

<table>
<thead>
<tr>
<th>Starter input files:</th>
<th>Engine file:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Modes computation</strong> and flexible-body input file</td>
<td></td>
</tr>
<tr>
<td>#---1-</td>
<td>-2-</td>
</tr>
<tr>
<td># EIGENMODES</td>
<td></td>
</tr>
<tr>
<td>#---1-</td>
<td>-2-</td>
</tr>
<tr>
<td>#/EIG/1</td>
<td></td>
</tr>
<tr>
<td># Elg 1</td>
<td></td>
</tr>
<tr>
<td># print Time beg Time end | Tracet</td>
<td></td>
</tr>
<tr>
<td>1192</td>
<td>1</td>
</tr>
<tr>
<td># Bnd</td>
<td>Start</td>
</tr>
<tr>
<td>15</td>
<td>0</td>
</tr>
<tr>
<td># Nbloc Incv Witer Ipri Tor</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>4</td>
</tr>
<tr>
<td>#---1-</td>
<td>-2-</td>
</tr>
</tbody>
</table>

| Explicit computation with flexible-body |
| \#---1-|-2-|-3-|-4-|-5-|-6-|-7-| |
| \# FLEXIBLE BODIES |
| \#---1-|-2-|-3-|-4-|-5-|-6-|-7-| |
| \#/FXBODY/1 |
| Flexible_body_Truck |
| \# Idmat 165209 |
| \# Filnam TRUCK_EIG_0001.xbd |
| \#---1-|-2-|-3-|-4-|-5-|-6-|-7-| |
| \# MASTER NODES FLEXIBLE BODIES |
| \#---1-|-2-|-3-|-4-|-5-|-6-|-7-| |
| \#/NODE 261200 -3.26715E+03 -1.17759E+01 1.407584E+03 |
| \#---1-|-2-|-3-|-4-|-5-|-6-|-7-| |

For the truck model, the global flexible body includes 14344 nodes, 120 of which are the master nodes of the inside rigid bodies. Thus, the flexible body takes into account constraints of the rigid bodies.
**Eigen Run**

In addition, you can define nine interface nodes linking the flexible body and the rest of the truck with the translation fixed along the X, Y and Z axis. Thus, 27 static modes will be computed.

Only the translation degrees are retained in order to minimize the input file size of the flexible body, given that preliminary studies have shown that additional static modes computed by fixing rotational degrees have not substantially improved flexible body behavior.

![Nine interface nodes with blocked translations for computing static modes.](image)

A static mode is computed for each fixed degree of freedom, in addition to the Eigen modes. Thus, the number of modes is equal to the number of Eigen modes, plus the number of blocked degrees of freedom.

**Flexible Body Run**

The rigid bodies and tied interfaces included in the flexible body domain should be removed for the second run. Those kinematic conditions are only considered in Eigen modes computation.

The coordinates of the center of mass (possible master node) indicated in the flexible body input file are:

X: \(3.267252\times10^3\)  
Y: \(-1.71759\times10^1\)  
Z: \(1.407584\times10^3\)  (node 265200)

The master node should be included in the nodes groups for gravity loading and initial velocity. It should be defined in the Starter file (NODE).

Connections between the parts covered by the flexible body and other parts of the model are modeled with beams and the rigid body, as shown in Fig 24. Connection is set at the beam extremity.

![Example of the connection point for flexible body.](image)
Simulation Results and Conclusions

Fig 25: Characteristic Eigen modes (arbitrary displacement).
Fig 26: Characteristic static modes (arbitrary displacement).
Comparison of Animation Results
Deformed configurations are compared with global bodies according to the modeling used:

Animation Results
- Animations multi models: cab deformation face view
- Animation flexible body model: cab deformation
- Animation original model: cab deformation

Conclusion
This example introduced a method for creating and employing a flexible body using an Eigen analysis performed by RADIOSS. The number of retained modes and the frequency range set for the Eigen analysis are according to the parameters which influenced the results.

Simulation using the flexible body provided accurate distribution of deformations in the model, compared with the modeling not having a substitute body. However, the amplitudes obtained are very low. The flexible body behavior could be enhanced by improving connections between the flexible body and the rest of the structure to ensure transmission of the shock wave up to the flexible body.

The flexible body input file required the IMPLICIT module (RADIOSS version 5) for the Eigen modes computation.