Created in COMSOL Multiphysics 6.1

Beam with a Traveling Load

Introduction

In some problems, the load changes position with time. It can be a train moving over a bridge or a heat source moving along a weld. In this example, intended to show how to model traveling loads, a set of load pulses travel along a beam with equally spaced supports.

The model is parameterized, so it is easy to change, for example, the velocity of the load pulses or the spacing between them. This is used for showing that for some load configurations, resonant vibrations will appear in the beam.

MODELING TRAVELING LOADS

The possibility to enter analytical expressions and use built-in or external functions in the input fields for loads and boundary conditions makes it easy model traveling loads in COMSOL Multiphysics. A load with a constant distribution moving with a constant velocity $\mathbf{v}$ can mathematically be written

$$f(\mathbf{x}-\mathbf{v}t) \tag{1}$$

This is the type of load being used in this example. You can, however, use completely arbitrary functions of space and time.

When the load is moving during the analysis, it will in general only cover some elements partially. This is not a problem, since the load is numerically integrated. The element size should, however, be chosen such that several elements are covered by the load.

Model Definition

GEOMETRY

Many aspects of the modeling can be parameterized. Table 1 below shows the values used in the analysis.

TABLE 1: GEOMETRICAL PARAMETERS.

PROPERTY	PARAMETER	VALUE IN MODEL
Number of spans	NumSpans	8
Length of individual spans	SpanWidth	10[m]
Beam Height	BeamHeight	0.3[m]

MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

The material is concrete, with material data taken from the Material Library:

- Young's modulus E = 25 GPa
- Poisson's ratio ν = 0.33
- Mass density ρ = 2300 kg/m^3

The beam is supported on one point at the end of each span.

The load travels along the top. The loading is controlled by the parameters listed in Table 2.

TABLE 2: LOAD PARAMETERS.

PROPERTY	PARAMETER	VALUES IN MODEL
Pressure under the load	LoadIntensity	0.1 MPa
Width of a load pulse	PulseWidth	1 m
Spacing between the load pulses	PulseSpacing	1: one pulse only 2: one pulse only 3: 2*SpanWidth 4: SpanWidth
Speed with which the load travels	LoadSpeed	1: 20 m/s 2: 89.7 m/s 3: 89.7 m/s 4: 89.7 m/s

As can be seen in the table, four different combinations of loads are analyzed:

1. A single load pulse traveling with a speed which is slow compared to the natural frequency of the beam.
2. A single loads pulse traveling with a speed which matches the natural frequency of the beam.
3. A pulse train traveling with a speed which matches the natural frequency of the beam. The distance between the individual load pulses is two span widths.
4. A pulse train traveling with a speed which matches the natural frequency of the beam. The distance between the individual load pulses is one span width.

Results

The lowest eigenfrequency of the beam can be computed analytically and is independent of the number of spans. It is actually the same as for a simply supported beam with the length of a single span. The expression is

$$f_0 = \frac{\pi}{2}\sqrt{\frac{EI}{\rho A L_s^4}} \tag{2}$$

which with the given parameters can be evaluated to 4.49 Hz. It can thus be expected that a load that travels one span L_s during half the period T_0 of the lowest natural frequency f_0 should be able to excite a resonant vibration. This critical velocity is then approximately

$$v_c = \frac{L_s}{\frac{T_0}{2}} = 2Lf_0 \approx 89.7 \text{ m/s} \tag{3}$$

The effect of changing the velocity of the load and the spacing between the loads is shown in Figure 1. The graphs show how the vertical displacement at the midpoint of the first span vary as function of the position of the first load pulse. The horizontal axis can also be seen as time, scaled with load velocity.

From the graphs, is can be seen that

- At a low load speed (20 m/s), the solution resembles a stationary solution. As the load moves away from the first span, the deflection there decreases.
- When the load speed is such (89.7 m/s) that the load travels two spans in the time T_0 of one period, then the load can actually excite resonances. This can be seen when using a very long load spacing (160 m, essentially giving a single load), as well as when using a spacing of two spans (20 m). The reason is that the load will always act on a downward motion of the beam, thus giving a positive power input.
- Even if the loads travel with the critical speed, they can counteract each other. This is the case in the last analysis, where the spacing between the loads matches the length of one span (10 m). In this case, every other load will act against the velocity, and thus limit the response.

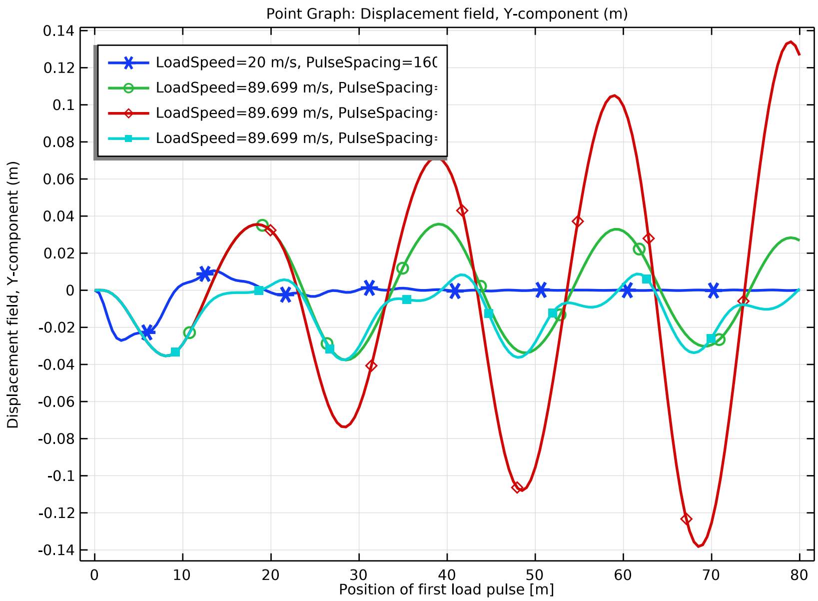

Figure 1: Displacement history at the midpoint of the first span, as function of the location of the first load pulse

A snapshot of the state when the loads

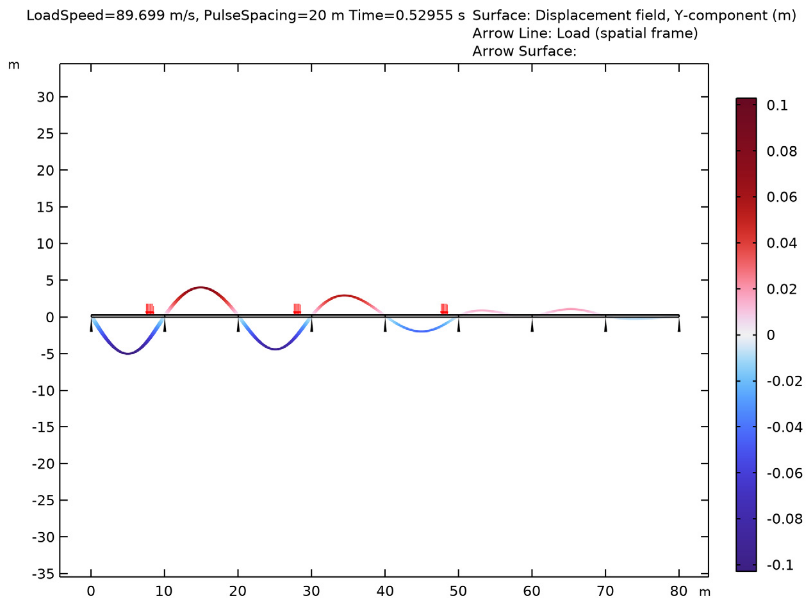

Figure 2: Displaced shape when loads are traveling to the right with a speed and spacing giving an amplification of the response.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/traveling_load`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `traveling_load_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `SpanWidth`.

4 In the **Height** text field, type `BeamHeight`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **y** text field, type `BeamHeight/2`.

Point 2 (pt2)

1 Right-click **Point 1 (pt1)** and choose **Duplicate**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `SpanWidth`.

4 Click **Build All Objects**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Union**, click **Build Selected**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `NumSpans`.

5 Locate the **Displacement** section. In the **x** text field, type `SpanWidth`.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

DEFINITIONS

Box: Points on Centerline

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Box: Points on Centerline` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Point**.

4 Locate the **Box Limits** section. In the **y minimum** text field, type `BeamHeight/3`.

5 In the **y maximum** text field, type `2*BeamHeight/3`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box: Boundaries Above

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Box: Boundaries Above` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y minimum** text field, type `2*BeamHeight/2`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Concrete**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Damping Settings** section.

3 In the β_{dK} text field, type `0.002`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Box: Points on Centerline**.

DEFINITIONS

Create functions describing the load. The distribution is a rectangular load.

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Parameters** section.

3 In the **Lower limit** text field, type `-0.5*PulseWidth`.

4 In the **Upper limit** text field, type `0.5*PulseWidth`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.1*PulseWidth`.

In order to make the load pulse periodic, call it through an analytical function, which has the option of being periodic.

Analytic 1 (an1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Pulse` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `rect1(x)`.

4 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
x	m

5 In the **Function** text field, type `1`.

6 Click to expand the **Periodic Extension** section. Select the **Make periodic** check box.

7 In the **Upper limit** text field, type `PulseSpacing`.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

Create a variable for filtering the load, so that it is periodic only behind the first pulse.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`FirstLoadX`	`LoadSpeed*t+PulseWidth/2`	m	`Position of the first load pulse`
`BehindFirstLoad`	`X<FirstLoadX`		`True if behind current position of the first load pulse`

Add the load definition to the whole upper boundary. The function call will make the load nonzero only at certain traveling positions.

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Box: Boundaries Above**.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	x
`if(BehindFirstLoad,-LoadIntensity*Pulse(X-LoadSpeed*t),0)`	y

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `1`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Box: Boundaries Above**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `ElemPerSpan`.

5 Click **Build All**.

Add a parametric sweep to study the four load pulse scenarios.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
LoadSpeed (Velocity of the load pulse)	`20[m/s] CriticalSpeed CriticalSpeed CriticalSpeed`
PulseSpacing (Distance between load pulses)	`TotLength*2 TotLength*2 SpanWidth*2 SpanWidth`

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,SpanWidth/LoadSpeed/20,Tend)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.0001`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type `Tstep`.

Set a reasonable scale for the displacements, so that the convergence checks in the solver will be appropriate.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 In the **Scale** text field, type `1e-2`.

Run the parametric sweep containing four time dependent studies.

9 In the **Study** toolbar, click **Compute**.

RESULTS

Displacement

In the **Settings** window for **2D Plot Group**, type `Displacement` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Displacement** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `v`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Wave>Wave** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

8 From the **Scale** list, choose **Linear symmetric**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `50`.

Modify the predefined load plot and add it on top of the displacement plot.

4 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Parametric Solutions 1 (sol2)>Solid Mechanics>Applied Loads (solid)>Boundary Loads (solid)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Load Arrows

1 In the **Model Builder** window, expand the **Results>Boundary Loads (solid)** node, then click **Boundary Load 1**.

2 In the **Settings** window for **Arrow Line**, type `Load Arrows` in the **Label** text field.

3 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Uniform**.

4 In the **Number of arrows** text field, type `1000`.

5 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

6 Select the **Scale factor** check box. In the associated text field, type `1.5e-5`.

7 In the **Boundary Loads (solid)** toolbar, click **Plot**.

8 In the **Model Builder** window, expand the **Load Arrows** node.

Color Expression, Deformation

1 In the **Model Builder** window, under **Results>Boundary Loads (solid)>Load Arrows**, Ctrl-click to select **Color Expression** and **Deformation**.

2 Right-click and choose **Delete**.

Load Arrows

In the **Model Builder** window, under **Results>Boundary Loads (solid)** right-click **Load Arrows** and choose **Copy**.

Load Arrows

In the **Model Builder** window, right-click **Displacement** and choose **Paste Arrow Line**.

Boundary Loads (solid)

In the **Model Builder** window, under **Results** right-click **Boundary Loads (solid)** and choose **Delete**.

Supports

1 In the **Model Builder** window, right-click **Displacement** and choose **Arrow Surface**.

   Add a visualization of the supports.

2 In the **Settings** window for **Arrow Surface**, type `Supports` in the **Label** text field.
3 Locate the **Expression** section. In the **X-component** text field, type `0`.
4 In the **Y-component** text field, type `1`.
5 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. From the **Entry method** list, choose **Coordinates**.
6 In the **Coordinates** text field, type `range(0,SpanWidth,TotLength)`.
7 Find the **Y grid points** subsection. From the **Entry method** list, choose **Coordinates**.
8 In the **Coordinates** text field, type `0`.
9 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
10 From the **Arrow base** list, choose **Head**.
11 Select the **Scale factor** check box. In the associated text field, type `2`.
12 From the **Color** list, choose **Black**.

Displacement

1 In the **Model Builder** window, click **Displacement**.
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3 From the **Time (s)** list, choose **0.52955**.
4 In the **Displacement** toolbar, click **Plot**.
5 From the **Parameter value (LoadSpeed (m/s),PulseSpacing (m))** list, choose **3: LoadSpeed=89.699 m/s, PulseSpacing=20 m**.
6 In the **Displacement** toolbar, click **Plot**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Animate the four different cases.

Animation 1

1 In the **Displacement** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Animation Editing** section.

3 From the **Parameter value (LoadSpeed (m/s),PulseSpacing (m))** list, choose **2: LoadSpeed=89.699 m/s, PulseSpacing=160 m**.

4 Click the **Play** button in the **Graphics** toolbar.

5 From the **Parameter value (LoadSpeed (m/s),PulseSpacing (m))** list, choose **3: LoadSpeed=89.699 m/s, PulseSpacing=20 m**.

6 Click the **Play** button in the **Graphics** toolbar.

7 From the **Parameter value (LoadSpeed (m/s),PulseSpacing (m))** list, choose **4: LoadSpeed=89.699 m/s, PulseSpacing=10 m**.

8 Click the **Play** button in the **Graphics** toolbar.

Create a point at the midpoint of the first span. This is where the vertical displacement will be compared between the four cases.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type `SpanWidth/2`.

4 In the **Y** text field, type `BeamHeight/2`.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

1D Plot Group 2

In the **Results** toolbar, click **1D Plot Group**.

Point Graph 1

1 Right-click **1D Plot Group 2** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `v`.

5 In the **1D Plot Group 2** toolbar, click **Plot**.

Since the time spans are different for different load speeds, it is more informative to study the results versus the position of the first load pulse.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `t*LoadSpeed`.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

10 From the **Positioning** list, choose **Interpolated**.

11 Click to expand the **Legends** section. Select the **Show legends** check box.

12 In the **1D Plot Group 2** toolbar, click **Plot**.

1D Plot Group 2

1 In the **Model Builder** window, click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Position of first load pulse [m]`.

6 In the **1D Plot Group 2** toolbar, click **Plot**.