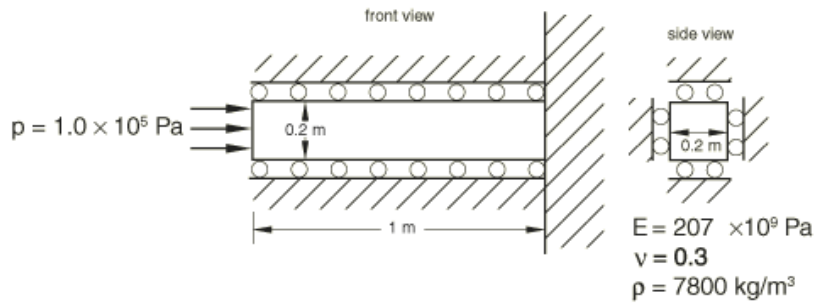


## 9.4 Example: stress wave propagation in a bar

This example demonstrates some of the fundamental ideas in explicit dynamics described earlier in [Chapter 2, “Abaqus Basics.”](#) It also illustrates stability limits and the effect of mesh refinement and material properties on the solution time.

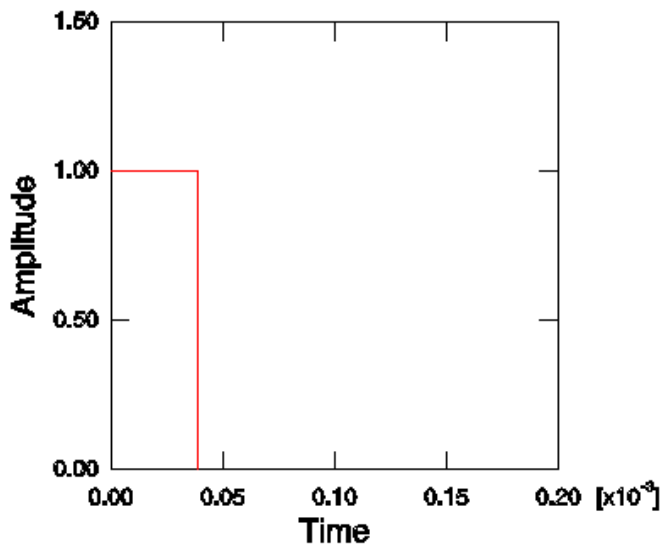
The bar has the dimensions shown in [Figure 9◆1](#).

**Figure 9◆1** Schematic for wave propagation in a bar.



To make the problem a one-dimensional strain problem, all four lateral faces are on rollers; thus, the three-dimensional model simulates a one-dimensional problem. The material is steel with the properties shown in [Figure 9◆1](#). The free end of the bar is subjected to a blast load with a magnitude of  $1.0 \times 10^5 \text{ Pa}$  and a duration of  $3.88 \times 10^{-5} \text{ s}$ . The normalized load versus time is shown in [Figure 9◆2](#).

**Figure 9◆2** Blast amplitude versus time.



### 9.4.1 Preprocessing—creating the model with Abaqus/CAE

In this section we discuss how to use Abaqus/CAE to create the model for this simulation. Abaqus provides scripts that replicate the complete analysis model for this problem. Run one of these scripts if you encounter difficulties following the instructions given below or if you wish to check your work. Scripts are available in the following locations:

- A Python script for this example is provided in [“Stress wave propagation in a bar,” Section A.7](#). Instructions on how to fetch

the script and run it within Abaqus/CAE are given in [Appendix A, “Example Files.”](#)

- A plug-in script for this example is available in the Abaqus/CAE Plug-in toolset. To run the script from Abaqus/CAE, select **Plug-ins** → **Abaqus** → **Getting Started**; highlight **Stress wave propagation in a bar**; and click **Run**. For more information about the Getting Started plug-ins, see “[Running the Getting Started with Abaqus examples,](#)” Section 82.1 of the [Abaqus/CAE User's Manual](#).

If you do not have access to Abaqus/CAE or another preprocessor, the input file that defines this problem can be created manually, as discussed in “[Example: stress wave propagation in a bar,](#)” Section 9.4 of [Getting Started with Abaqus: Keywords Edition](#).

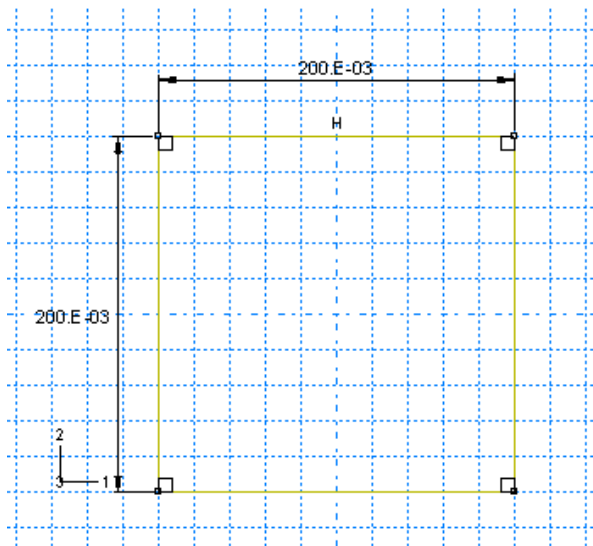
### Defining the model geometry

In this example you will create a three-dimensional, deformable body with a solid, extruded base feature. You will first sketch the two-dimensional profile of the bar and then extrude it.

#### To create a part:

1. Create a new part named `Bar`, and accept the default settings of a three-dimensional, deformable body and a solid, extruded base feature in the **Create Part** dialog box. Use an approximate size of 0.5 for the model.
2. Use the dimensions given in [Figure 9-3](#) to sketch the cross-section of the bar.

**Figure 9-3** Rectangular cross-section (with grid spacing doubled).



The following possible approach can be used:

- a. Create a rectangle centered about the origin using the **Create Lines: Rectangle** tool.

**Tip:** Create fixed horizontal and vertical construction lines through the origin of the sketch plane, and apply appropriate geometry constraints to the sketch.

- b. Dimension the sketch so that the cross-section is 0.20 m high × 0.20 m wide.
- c. Click **Done** in the prompt area when you are finished sketching the profile.

The **Edit Base Extrusion** dialog box appears. To complete the part definition, you must specify the distance over which the profile will be extruded.

- d. In the dialog box, enter an extrusion depth of 1.0 m.
3. Save your model in a model database file named `Bar.cae`.

### Defining the material and section properties

Create a single linear elastic material named **Steel** with a mass density of  $7800 \text{ kg/m}^3$ , a Young's modulus of  $207\text{E}9 \text{ Pa}$ , and a Poisson's ratio of  $0.3$ .

Create a solid, homogeneous section definition named **BarSection** with **Steel** as the material.

Assign the section definition **BarSection** to the entire part.

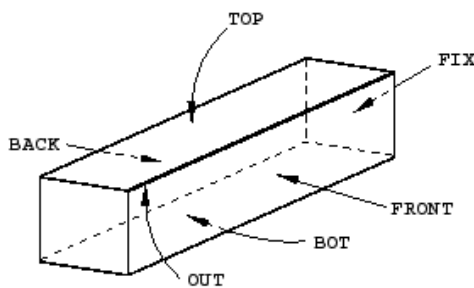
### Creating an assembly

Create an instance of the bar. The model is oriented by default so that the global 3-axis lies along the length of the bar.

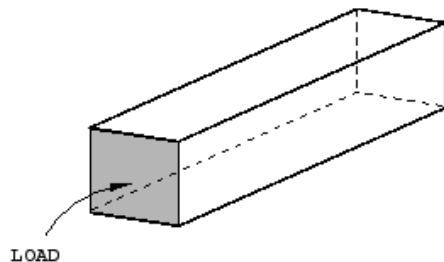
### Creating geometry sets and surfaces

Create the geometry sets **TOP**, **BOT**, **FRONT**, **BACK**, **FIX**, and **OUT** as shown in [Figure 9-4](#). (The set **OUT** contains the edge shown in bold in [Figure 9-4](#).) Create the surface named **LOAD** shown in [Figure 9-5](#). These regions will be used later for the application of loads and boundary conditions, as well as for the definition of output requests.

**Figure 9-4** Sets.



**Figure 9-5** Surface.



### Defining steps

Create a single dynamic, explicit step named **BlastLoad**. Enter **Apply pressure load pulse** as the step description, and set the **Time period** to  $2.0\text{E}-4 \text{ s}$ . In the **Edit Step** dialog box, click the **Other** tab. To keep the stress wave as sharp as possible, set the **Quadratic bulk viscosity parameter** (discussed in [“Bulk viscosity,” Section 9.5.1](#)) to zero.

### Specifying output requests

Edit the default field output request so that preselected field data are written to the output database at four equally spaced intervals for the step **BlastLoad**.

Delete the existing default history output request, and create a new set of history output requests. In the **Create History Output** dialog box, accept the default name of **H-Output-1** and select the **BlastLoad** step. Click **Continue**. Click the arrow next to the **Domain** field, select **Set**, and select **OUT**. In the **Output Variables** section, click on the triangle to the left of **Stresses**. Click on the triangle to the left of **S, Stress components and invariants**, and toggle on the **S33** variable, which is the component of stress in the axial direction of the bar. Specify that output be saved at every increment.

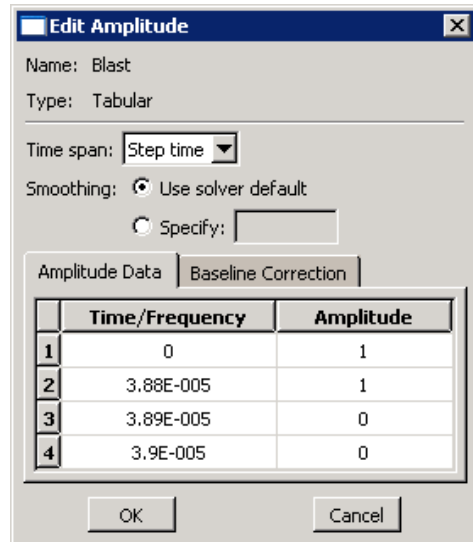
### Prescribing boundary conditions

Create a boundary condition named `Fix right end`, and constrain the right end of the bar (geometry set `FIX`) in all three directions (see [Figure 9◆1](#)). Create additional boundary conditions to constrain the top, bottom, front, and back faces in the directions normal to these faces (the 1-direction for sets `FRONT` and `BACK` and the 2-direction for sets `TOP` and `BOT`).

### Defining the load history

The blast load is applied at its maximum value instantaneously and is held constant for  $3.88 \times 10^{-5}$  s. Then the load is suddenly removed and held constant at zero. Create an amplitude definition named `Blast` using the data shown in [Figure 9◆6](#). For this problem the pressure load at any given time is the specified magnitude of the pressure load times the value interpolated from the amplitude curve.

**Figure 9◆6** Tabular data for the blast load amplitude definition.



Create a pressure load named `Blast load`, and select `BlastLoad` as the step in which it will be applied. Apply the load to the surface `LOAD`. Select **Uniform** for the distribution, specify a value of  $1.0 \text{E}5$  Pa for the load magnitude, and select **Blast** for the amplitude.

### Creating a mesh

Using the material properties (neglecting Poisson's ratio), we can calculate the wave speed of the material using the equations introduced earlier.

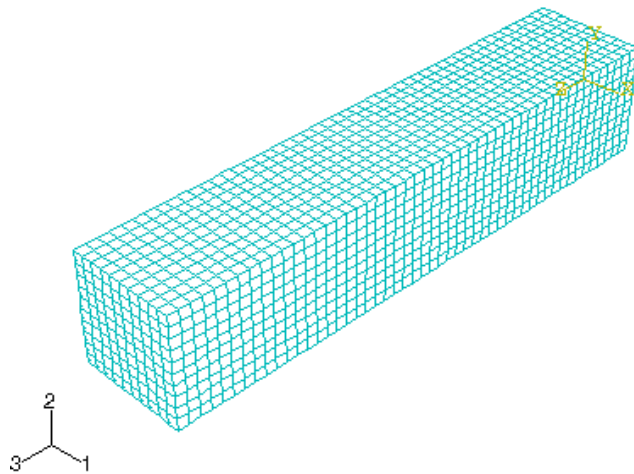
$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{207 \times 10^9 \text{ Pa}}{7800 \text{ kg/m}^3}} = 5.15 \times 10^3 \text{ m/s.}$$

At this speed the wave passes to the fixed end of the bar in  $1.94 \times 10^{-4}$  s. Since we are interested in stress propagation along the length of the bar through time, we need a sufficiently refined mesh to capture the stress wave accurately. We will assume that the blast load will take place over the span of 10 elements. To determine the length of these 10 elements, multiply the blast duration by the wave speed:

$$L_{10el} = (3.88 \times 10^{-5} \text{ s}) c_d.$$

The length of 10 elements is 0.2 m. Since the total length of the bar is 1.0 m, we would have 50 elements along the length. To keep the mesh uniform, we will also have 10 elements in each of the transverse directions, making the mesh  $50 \times 10 \times 10$ . This mesh is shown in [Figure 9◆7](#).

**Figure 9◆7**  $50 \times 10 \times 10$  mesh.



Seed the part with a target global element size of 0.02. Select C3D8R from the **Explicit** library as the element type, and mesh the part.

**Note:** The proposed mesh density exceeds the model size limits of the Abaqus Student Edition. If you are using this product, specify one of the following:

- A global element size of 0.04.
- A 50  $\times$  3  $\times$  3 mesh.
- A 25  $\times$  5  $\times$  5 mesh.

### Creating, running, and monitoring a job

Create a job named `Bar`, and enter `Stress wave propagation in a bar (SI units)` for the job description. Submit the job, and monitor the results. If any errors are encountered, correct the model and rerun the simulation. Be sure to investigate the cause of any warning messages and take appropriate action; recall that some warning messages can be ignored safely while others require corrective action.

### The status (.sta) file

You can also look at the status file, `Bar.sta`, to monitor the job progress. You can browse or search this file by clicking the **Status File** tab in the **Job Monitor**. The status file contains information about moments of inertia, followed by information concerning the stability limit. The 10 elements with the lowest stable time limits are listed in rank order.

```

Most critical elements:
Element number   Rank   Time increment   Increment ratio
(Instance name)
-----
      1           1   1.819458E-06   1.000000E+00
BAR-1
     10           2   1.819458E-06   1.000000E+00
BAR-1
     21           3   1.819458E-06   1.000000E+00
BAR-1
     30           4   1.819458E-06   1.000000E+00
BAR-1
     31           5   1.819458E-06   1.000000E+00
BAR-1
     40           6   1.819458E-06   1.000000E+00
BAR-1
     51           7   1.819458E-06   1.000000E+00

```

```

BAR-1
    60      8      1.819458E-06      1.000000E+00
BAR-1
    61      9      1.819458E-06      1.000000E+00
BAR-1
    70     10      1.819458E-06      1.000000E+00
BAR-1

```

The status file continues with information about the progress of the solution.

```
STEP 1 ORIGIN 0.0000
```

```

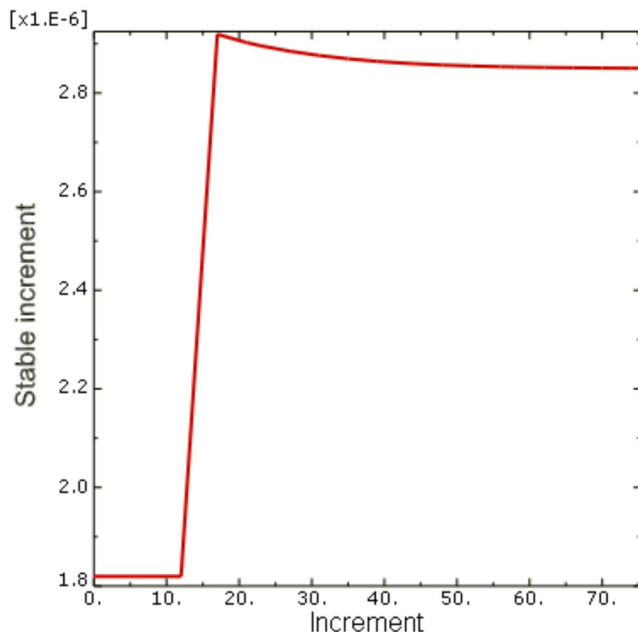
Total memory used for step 1 is approximately 6.7 megabytes.
Global time estimation algorithm will be used.
Scaling factor : 1.0000
Variable mass scaling factor at zero increment: 1.0000

```

INCREMENT	STEP TIME	TOTAL TIME	CPU TIME	STABLE INCREMENT	CRITICAL ELEMENT	KINETIC ENERGY	TOTAL ENERGY
0	0.000E+00	0.000E+00	00:00:00	1.819E-06	1	0.000E+00	0.000E+00
INSTANCE WITH CRITICAL ELEMENT: BAR-1							
ODB Field Frame Number 0 of 4 requested intervals at increment zero.							
6	1.092E-05	1.092E-05	00:00:00	1.819E-06	1	4.432E-05	-1.134E-06
12	2.183E-05	2.183E-05	00:00:00	1.819E-06	21	9.043E-05	-1.437E-06
17	3.431E-05	3.431E-05	00:00:00	2.919E-06	1	1.424E-04	-1.443E-06
21	4.596E-05	4.596E-05	00:00:00	2.902E-06	1	1.596E-04	1.744E-06
23	5.176E-05	5.176E-05	00:00:00	2.896E-06	1	1.581E-04	3.095E-06
ODB Field Frame Number 1 of 4 requested intervals at 5.176230E-05							
27	6.333E-05	6.333E-05	00:00:00	2.885E-06	1	1.565E-04	1.089E-06
31	7.485E-05	7.485E-05	00:00:00	2.876E-06	121	1.550E-04	-4.690E-08

Similar information is available in the **Job Diagnostics** dialog box of the Visualization module. This dialog box also allows you to plot histories of the data, as shown in [Figure 9◆8](#). To create a history plot, select a step from the **Job History**, select a column in the **Incrementation** tab, and click **Plot selected column**.

**Figure 9◆8** Stable time increment history.



## 9.4.2 Postprocessing

In the Model Tree, click mouse button 3 on the job named **Bar** and select **Results** from the menu that appears to enter the Visualization module and automatically open the output database (.odb) file created by this job. Alternatively, from the **Module** list located in the context bar, select **Visualization** to enter the Visualization module; open the .odb file by selecting **File**→**Open** from the main menu bar and double-clicking on the appropriate file.

### Plotting the stress along a path

We are interested in looking at how the stress distribution along the length of the bar changes with time. To do so, we will look at the stress distribution at three different times throughout the course of the analysis.

Create a curve of the variation of the stress in the 3-direction (S33) along the axis of the bar for each of the first three frames of the output database file. To create these plots, you first need to define a straight path along the axis of the bar.

#### To create a point list path along the center of the bar:


1. In the Results Tree, double-click **Paths**.

The **Create Path** dialog box appears.

2. Name the path **Center**. Select **Point list** as the path type, and click **Continue**.

The **Edit Point List Path** dialog box appears.

3. In the **Point Coordinates** table, enter the coordinates of the centers of both ends of the bar. The input specifies a path from the first point to the second point, as defined in the global coordinate system of the model.

**Note:** If you generated the geometry and mesh using the procedure described earlier, the table entries are 0, 0, 1 and 0, 0, 0. If you used an alternate procedure to generate the bar geometry, you can use the  tool in the **Query** toolbar to determine the coordinates of the centers at each end of the bar.

4. When you have finished, click **OK** to close the **Edit Point List Path** dialog box.

#### To save X–Y plots of stress along the path at three different times:

1. In the Results Tree, double-click **XYData**.

The **Create XY Data** dialog box appears.

2. Choose **Path** as the X–Y data source, and click **Continue**.

The **XY Data from Path** dialog box appears with the path that you created visible in the list of available paths. If the undeformed model shape is currently displayed, the path you select is highlighted in the plot.

3. Toggle on **Include intersections** under **Point Locations**.
4. Accept **True distance** as the selection in the **X Values** region of the dialog box.
5. Click **Field Output** in the **Y Values** region of the dialog box to open the **Field Output** dialog box.
6. Select the S33 stress component, and click **OK**.

The field output variable in the **XY Data from Path** dialog box changes to indicate that stress data in the 3-direction (S33) will be created.

**Note:** Abaqus/CAE may warn you that the field output variable will not affect the current image. Leave the plot state **As is**, and click **OK** to continue.

7. Click **Step/Frame** in the **Y Values** region of the **XY Data from Path** dialog box.
8. In the **Step/Frame** dialog box that appears, choose frame 1, which is the second of the five recorded frames. (The first frame listed, frame 0, is the base state of the model at the beginning of the step.) Click **OK**.

The **Y Values** region of the **XY Data from Path** dialog box changes to indicate that data from Step 1, frame 1 will be created.

- To save the  $X$ - $Y$  data, click **Save As**.

The **Save XY Data As** dialog box appears.

- Name the  $X$ - $Y$  data S33\_T1, and click **OK**.


S33\_T1 appears in the **XYData** container of the Results Tree.

- Repeat Steps 7 through 9 to create  $X$ - $Y$  data for frames 2 and 3. Name the data sets S33\_T2 and S33\_T3, respectively.
- To close the **XY Data from Path** dialog box, click **Cancel**.

#### To plot the stress curves:

- In the **XYData** container, drag the cursor to select all three  $X$ - $Y$  data sets.
- Click mouse button 3, and select **Plot** from the menu that appears.

Abaqus/CAE plots the stress in the 3-direction along the center of the bar for frames 1, 2, and 3, corresponding to approximate simulation times of  $5 \times 10^{-5}$  s,  $1 \times 10^{-4}$  s, and  $1.5 \times 10^{-4}$  s, respectively.

- Click  in the prompt area to cancel the current procedure.

#### To customize the $X$ - $Y$ plot:

- Double-click the  $Y$ -axis.


The **Axis Options** dialog box appears. The **Y Axis** is selected.

- In the **Tick Mode** region of the **Scale** tabbed page, select **By increment** and specify that the  $Y$ -axis major tick marks occur at  $20E3$  Pa increments.

You can also customize the axis titles.

- Switch to the **Title** tabbed page.
- Enter `Stress - S33 (Pa)` as the  $Y$ -axis title.
- To edit the  $X$ -axis, select the axis label in the **X Axis** field of the dialog box. In the **Title** tabbed page of the dialog box, enter `Distance along bar (m)` as the  $X$ -axis title.
- Click **Dismiss** to close the **Axis Options** dialog box.

#### To customize the appearance of the curves in the $X$ - $Y$ plot:

- In the Visualization toolbox, click  to open the **Curve Options** dialog box.
- In the **Curves** field, select S33\_T2.
- Choose the dotted line style for the S33\_T2 curve.

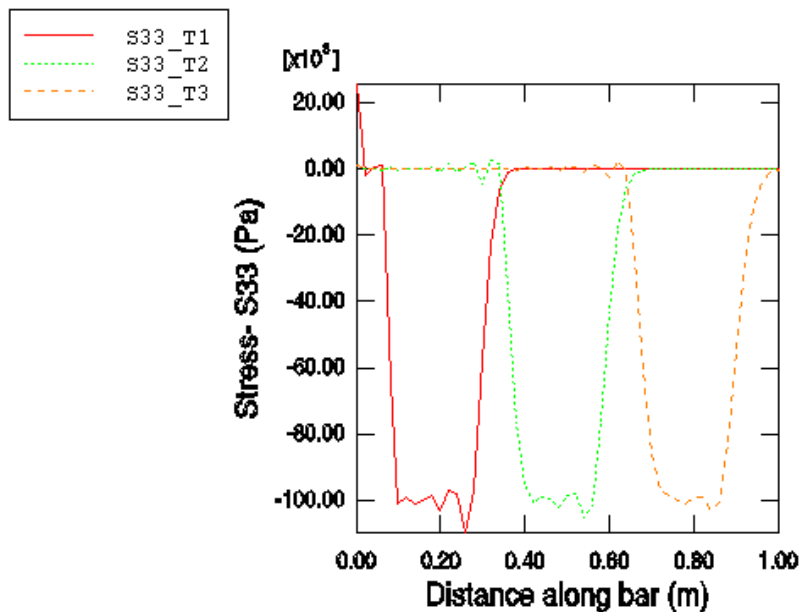
The S33\_T2 curve becomes dotted.

- Repeat Steps 2 and 3 to make the S33\_T3 curve dashed.
- Dismiss the **Curve Options** dialog box.

The customized plot appears in [Figure 9-9](#). (For clarity, the default grid and legend positions have been changed.)



**Figure 9◆9** Stress (S33) along the bar at three different time instances.




We can see that the length of the bar affected by the stress wave is approximately 0.2 m in each of the three curves. This distance should correspond to the distance that the blast wave travels during its time of application, which can be checked by a simple calculation. If the length of the wave front is 0.2 m and the wave speed is  $5.15 \times 10^3$  m/s, the time it takes for the wave to travel 0.2 m is  $3.88 \times 10^{-5}$  s. As expected, this is the duration of the blast load that we applied. The stress wave is not exactly square as it passes along the bar. In particular, there is “ringing” or oscillation of the stress behind the sudden changes in stress. Linear bulk viscosity, discussed later in this chapter, damps the ringing so that it does not affect the results adversely.

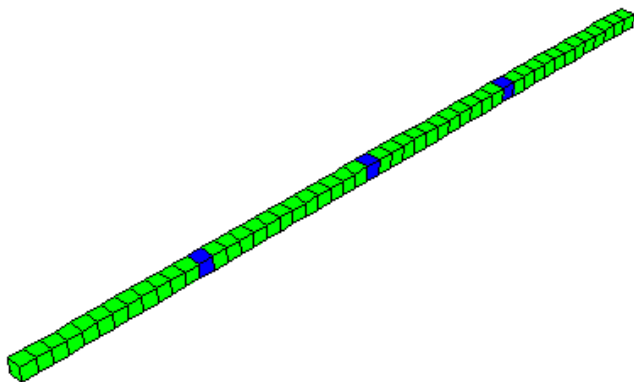
### Creating a history plot

Another way to study the results is to view the time history of stress at three different points within the bar; for example, at distances of 0.25 m, 0.50 m, and 0.75 m from the loaded end of the bar. To do this, we must first determine the labels of the elements located at these positions. An easy way to accomplish this is to query the elements in a display group consisting of the elements along the edge of the bar (set OUT).


#### To query the element labels:

1. In the Results Tree, expand the **Element Sets** container underneath the output database file named `Bar.odb`. Click mouse button 3 on the set named **OUT**, and select **Replace** from the menu that appears.
2. From the main menu bar, select **Tools**→**Query**; or use the  tool in the **Query** toolbar.
3. In the **Query** dialog box that appears, click **Element**.
4. Click on the elements shaded in [Figure 9◆10](#) (every 13th element along the bar). The element ID (label) appears in the prompt area (and also in the message area). Make note of the element labels for the three shaded elements.

**Figure 9◆10** History plot display group.



### To plot the stress history:


1. In the Results Tree, click mouse button 3 on **History Output** and deselect **Group Children** from the menu that appears.
2. Select the data for the three elements you have identified (again, every 13th element). Use **[Ctrl]+Click** to select multiple  $X$ - $Y$  data sets.
3. Click mouse button 3, and select **Plot** from the menu that appears.  
Abaqus/CAE displays an  $X$ - $Y$  plot of the longitudinal stress in each element versus time.
4. Click  in the prompt area to cancel the current procedure.

As before, you can customize the appearance of the plot.

### To customize the $X$ - $Y$ plot:

1. Double-click the  $X$ -axis.  
The **Axis Options** dialog box appears.
2. Switch to the **Title** tabbed page.
3. Specify `Total time (s)` as the  $X$ -axis title.
4. Click **Dismiss** to close the dialog box.

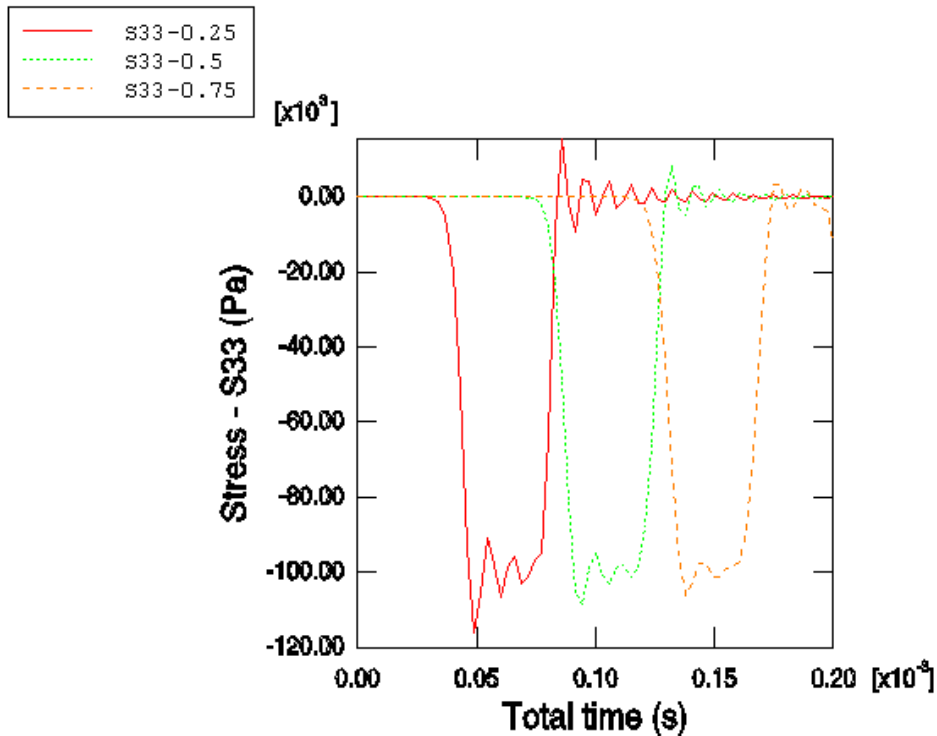
### To customize the appearance of the curves in the $X$ - $Y$ plot:

1. In the Visualization toolbox, click  to open the **Curve Options** dialog box.
2. In the **Curves** field, select the temporary  $X$ - $Y$  data label that corresponds to the element closest to the free end of the bar. (Of the elements in this set, this one is affected first by the stress wave.)
3. Enter `S33-0.25` as the curve legend text.
4. In the **Curves** field, select the temporary  $X$ - $Y$  data label that corresponds to the element in the middle of the bar. (This is the element affected next by the stress wave.)
5. Specify `S33-0.5` as the curve legend text, and change the curve style to dotted.
6. In the **Curves** field, select the temporary  $X$ - $Y$  data label that corresponds to the element closest to the fixed end of the bar. (This is the element affected last by the stress wave.)
7. Specify `S33-0.75` as the curve legend text, and change the curve style to dashed.

8. Click **Dismiss** to close the dialog box.

The customized plot appears in [Figure 9◆11](#). (For clarity, the default grid and legend positions have been changed.)

**Figure 9◆11** Time history of stress (S33) at three points along the length of the bar (0.25 m, 0.5 m, and 0.75 m).

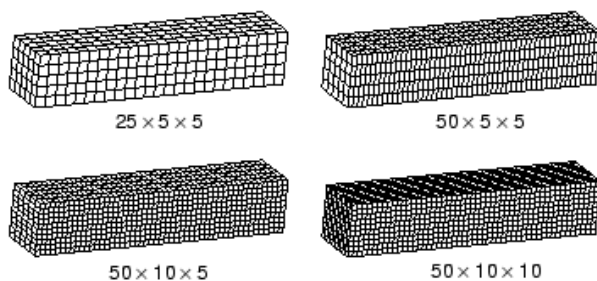


In the history plot we can see that stress at a given point increases as the stress wave travels through the point. Once the stress wave has passed completely through the point, the stress at the point oscillates about zero.

### 9.4.3 How the mesh affects the stable time increment and CPU time

In “[Automatic time incrementation and stability](#),” [Section 9.3](#), we discussed how mesh refinement affects the stability limit and the CPU time. Here we will illustrate this effect with the wave propagation problem. We began with a reasonably refined mesh of square elements with 50 elements along the length and 10 elements in each of the two transverse directions. For illustrative purposes, we will now use a coarse mesh of 25 ◆ 5 ◆ 5 elements and observe how refining the mesh in the various directions changes the CPU time. The four meshes are shown in [Figure 9◆12](#).

**Figure 9◆12** Meshes from least to most refined.



[Table 9◆1](#) shows how the CPU time (normalized with respect to the coarse mesh model result) changes with mesh refinement for this problem. The first half of the table provides the expected results, based on the simplified stability equations presented in this guide; the second half of the table provides the results obtained by running the analyses in Abaqus/Explicit on a desktop workstation.

**Table 9◆1** Mesh refinement and solution time.

Mesh	Simplified Theory			Actual		
	$\Delta t_{\text{stable}}$ (s)	Number of Elements	CPU Time (s)	Max $\Delta t_{\text{stable}}$ (s)	Number of Elements	Normalized CPU Time
25 ◆ 5 ◆ 5	A	B	C	6.06e-6	625	1
50 ◆ 5 ◆ 5	A/2	2B	4C	3.14e-6	1250	4
50 ◆ 10 ◆ 5	A/2	4B	8C	3.12e-6	2500	8.33
50 ◆ 10 ◆ 10	A/2	8B	16C	3.11e-6	5000	16.67

For the theoretical results we choose the coarsest mesh, 25 ◆ 5 ◆ 5, as the base state, and we define the stable time increment, the number of elements, and the CPU time as variables A, B, and C, respectively. As the mesh is refined, two things happen: the smallest element dimension decreases, and the number of elements in the mesh increases. Each of these effects increases the CPU time. In the first level of refinement, the 50 ◆ 5 ◆ 5 mesh, the smallest element dimension is cut in half and the number of elements is doubled, increasing the CPU time by a factor of four over the previous mesh. However, further doubling the mesh to 50 ◆ 10 ◆ 5 does not change the smallest element dimension; it only doubles the number of elements. Therefore, the CPU time increases by only a factor of two over the 50 ◆ 5 ◆ 5 mesh. Further refining the mesh so that the elements are uniform and square in the 50 ◆ 10 ◆ 10 mesh again doubles the number of elements and the CPU time.

This simplified calculation predicts quite well the trends of how mesh refinement affects the stable time increment and CPU time. However, there are reasons why we did not compare the predicted and actual stable time increment values. First, recall that we made the approximation that the stable time increment is

$$\Delta t_{\text{stable}} = \frac{L^e}{c_d}$$

We then assumed that the characteristic element length,  $L^e$ , is the smallest element dimension, whereas Abaqus/Explicit actually determines the characteristic element length based on the overall size and shape of the element. Another complication is that Abaqus/Explicit employs a global stability estimator, which allows a larger stable time increment to be used. These factors make it difficult to predict the stable time increment accurately before running the analysis. However, since the trends follow nicely from the simplified theory, it is straightforward to predict how the stable time increment will change with mesh refinement.

#### 9.4.4 How the material affects the stable time increment and CPU time

The same wave propagation analysis performed on different materials would take different amounts of CPU time, depending on the wave speed of the material. For example, if we were to change the material from steel to aluminum, the wave speed would change from  $5.15 \times 10^3$  m/s to

$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{70 \times 10^9 \text{ Pa}}{2700 \text{ kg/m}^3}} = 5.09 \times 10^3 \text{ m/s.}$$

The change from aluminum to steel has minimal effect on the stable time increment, because the stiffness and the density differ by nearly the same amount. In the case of lead the difference is more substantial, as the wave speed decreases to

$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{14 \times 10^9 \text{ Pa}}{11240 \text{ kg/m}^3}} = 1.12 \times 10^3 \text{ m/s,}$$

which is approximately one-fifth the wave speed of steel. The stable time increment for the lead bar would be five times the stable time increment of our steel bar.