BEDAS 4.0

MechaTools Technologies

User's Guide

BEDAS User's Guide Copyright © 2004 MechaTools Technologies, Inc. All rights reserved. Beam Designer™ is trademark of MechaTools Technologies.

The software described in this documentation is furnished under a license agreement and may be used only in accordance with the terms of the agreement.

DISCLAIMER

Trademarks

LICENSE & COPY RESTRICTIONS
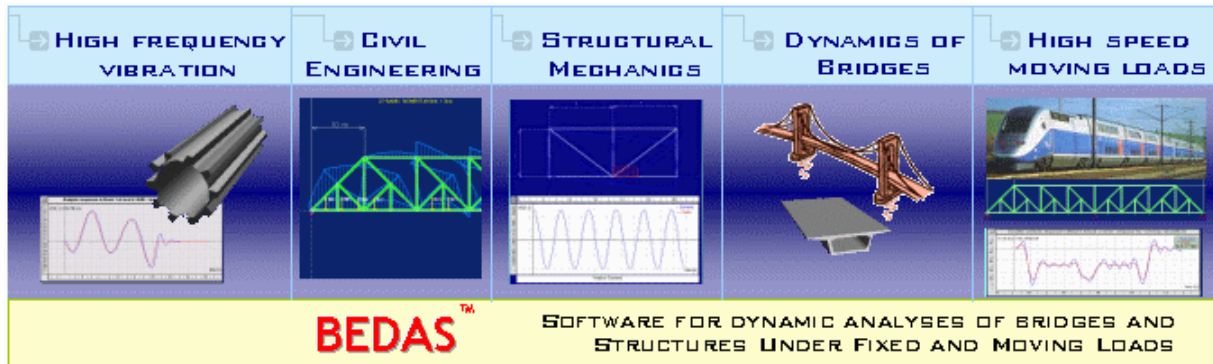
MechaTools Technologies – 3030, Louvigny, Sainte-Foy, Quebec - G1W1B1 Canada.
Technical Support: support@mechatools.com
Phone: (418) 651.3689
Fax: (418) 651.6456
Information: info@mechatools.com
www.mechatools.com

About This Manual

This manual describes **BEDAS**^{TM} a professional level bridge dynamic analysis under moving loads program, with capabilities to handle most typical 2d mechanics, aeronautics and civil/structural analysis problems. It performs static, mode shapes and dynamic response analysis. It gives exact high frequencies and mode shapes with only one element per beam (span). Dynamic finite elements lead to higher precision of internal forces in any location of structure excited by fixed, moving and/or seismic loads.
 BEDAS is a stand-alone application with many advanced engineering functionalities.

This manual is divided in six chapters:

Chapter 1 Installation

This chapter describes system requirements and gets you started installing BEDAS TM software.

Chapter 2 Overview of BEDAS environment

This chapter presents the environment and basic functionalities of Beam Designer.

Chapter 3 Step-by-step menus

This chapter presents a step-by-step example to show most functionality of BEDAS from starting the application to printing results.

Chapter 4 Commands and Toolbars

This chapter presents an overview of the operations of BEDAS and a summary of the commands used.

Chapter 5 Components and Tips

Using practical examples, this chapter presents an overview of the operations of BEDAS components and tips used in sketching and designing beam structures.

Chapter 6 Techniques and conventions

This chapter discusses the numerical methods implemented in BEDAS followed by some interesting examples.

Table of Contents

Chapter 1

Installation

System Requirements

BEDAS runs under Windows 98/Me/NT/2000/XP. It requires a Pentium class CPU or better and a minimum of 64 MB memory is recommended. The software requires 20 MB of free space on your hard disk.

Installing Beam Designer

To install Beam Designer, run the **setup.exe** program located on the installation CD.
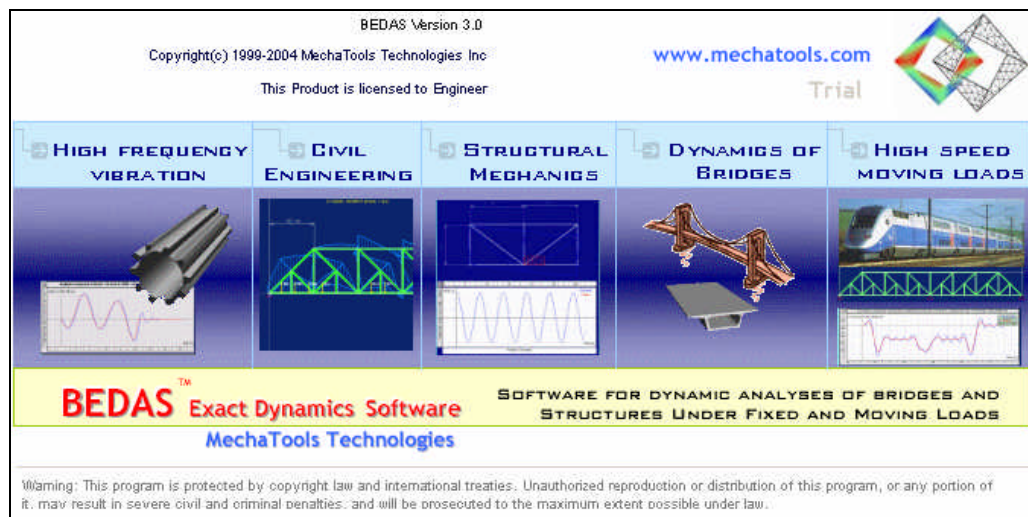
License Types

After BEDAS files are installed, the final step in the installation is to inform BEDAS of your licensing information as printed on your invoice. The licensing information consists of a Name, either your name or the name of your organization or company, and a coded key. This information should be entered in the login dialog presented when you run the application for the first time. The software will check to see if the key is valid and if so, will record the information on your computer. BEDAS software is licensed either to individual engineers or to their companies or organizations. The license agreement is included with the software when you download or purchase it, and is posted on our web site **www.mechatools.com**. Essentially, the **single-user license** agreement states that you may use the software **on one single machine**, but that you must purchase additional licenses to allow others to use the software, on other machines. For more information, please contact Mechatools Technologies.

Running BEDAS

By default, the BEDAS installation program will create menu items in the Program Files directory. To run the program, choose **Start | Program | MechaTools Studio| BEDAS**.

Mechatools Team

BEDAS is the easiest way for engineers to efficiently perform the dynamic responses of bridges under moving and fixed loads. BEDAS was developed by a professionals and engineers at Mechatools Technologies. This product was developed using state of the art methods in exact dynamic finite elements and sophisticated algorithms. BEDAS is robust, scalable and very easy to use.

How to Contact MechaTools Technologies Support

The fast and efficient way to contact MechaTools for technical support is through email. Our technical assistance address is **support@mechatools.com**. For reporting problems, you may call or fax our office as well. Our office hours are 8:30AM - 5:00PM Eastern Time US & Canada (GMT-05:00). The technical support phone number is (**418) 651.3689** and fax line is (**418). 651.6456**.

Chapter 2

BEDAS Environment

Environment

The BEDAS environment is shown in the picture below.
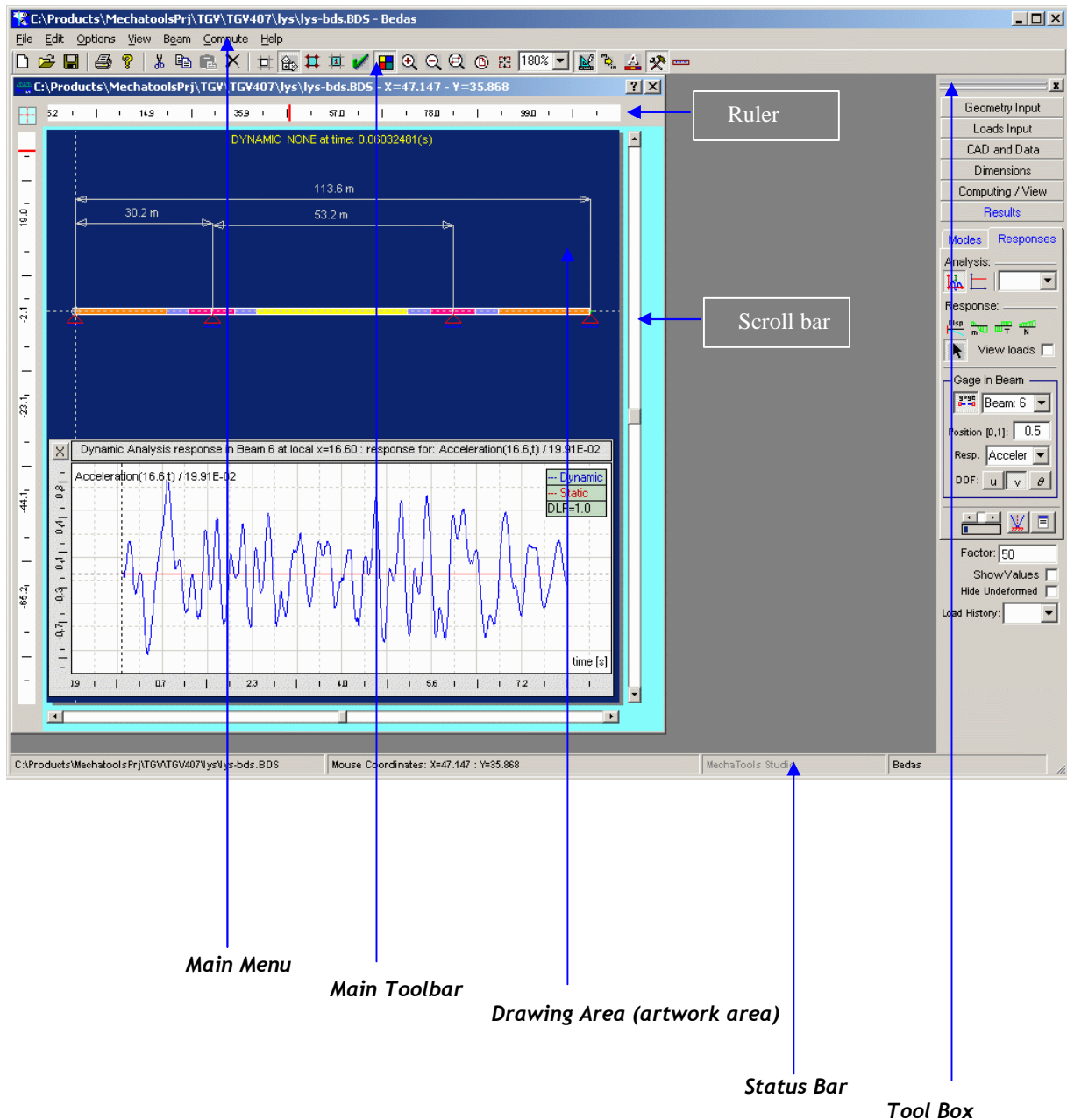
MechaTools Studio is a working environment that supports various engineering tools. When you install a stand-alone version of BEDAS, you actually install MechaTools Studio with one engineering tool: BEDAS.

The main menu, toolbars, and status bar are standard Windows components. You can toggle the display of toolbars using commands in the View menu. The toolbox may also be "undocked" and moved in the program window from left to right by pressing right mouse button. The status bar at the bottom provides project information, hints and messages. It displays the current filename, the mouse position and the centroïd of the selected beams.

Chapter 3

Step-By-Step menus

Starting BEDAS

Start BEDAS with the **Start | Programs | BEDAS** menu.

Creating a New Project

To create a new structure project, click the **New Project** icon from the toolbar or click **New** from the **File** menu.

The **Save As** dialog is automatically displayed. Enter the name of you new structure project and click save. This dialog may be invoked at any time using the **File** menu.

Setting the New Project Units

The **Units Transformation** dialog is automatically displayed. Enter the units you will use to define the new structure. These units will be used for all work in Beam Designer. The **Units Transformation** dialog may be invoked at any time by clicking the **Change Units** icon from the toolbar or from the **Options** menu by clicking **Change Units**. Changing units does NOT change dimensions. A 1-inch edge becomes a 2.54 cm edge. Click **OK** when you are done.
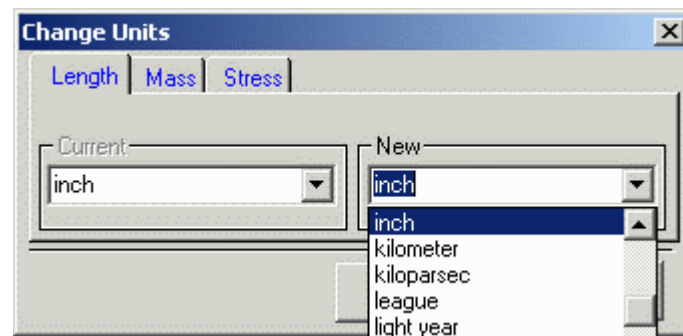
Setting the Bounding Box

The **Working Window Boundary** dialog is automatically displayed. This dialog is used to change the size and position of the bounding box. The bounding box is an imaginary box that surrounds a region of the drawing. Zooming operations are based on the size and position of the bounding box. For example, the **Zoom All** command resizes and moves the viewport so that the bounding box is centered and fully visible in the drawing area. After the **Zoom All** command is executed, the **Zoom** dropdown is set to 100%. BEDAS automatically resizes the bounding box when a beam manipulation causes a beam to exceed the bounding box limits. The user may change the bounding box manually to concentrate on a specific region of the drawing. The **Bounding Box** dialog may be invoked at any time by clicking the **Changing Bounding Box by Mouse** or the **Changing Bounding Box Manually** icons from the **CAD and Data** tab of the toolbox **Options** menu by clicking **Change Bounding Box | Manually**. Click **OK** when you are done.
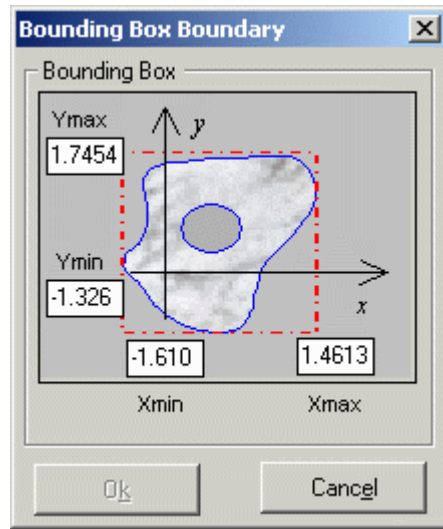
Setting the Project Preference

The **Preferences** dialog is automatically displayed. This dialog is used to associate structure specific information to the new structure project. Click the **Identification** tab to edit the project information. Generally, you may accept the default values for the **Colors**, **Fonts** and **General** tabs.

The **Color** tab is used to set the color of various display elements:
a) Beam identifiers
b) Beam labels
c) Point identifiers
d) Grid
e) Dimensions
f) Dynamic responses visualization

The **Font** tab is used to set the font of various display elements:
a) Beam identifiers
b) Beam labels
c) Point identifiers
d) Dimensions

The **preferences** dialog may be invoked at any time by clicking on the **Preference**s icon of the toolbar or from the **Options** menu by clicking **Preferences**. Click **OK** when you are done.
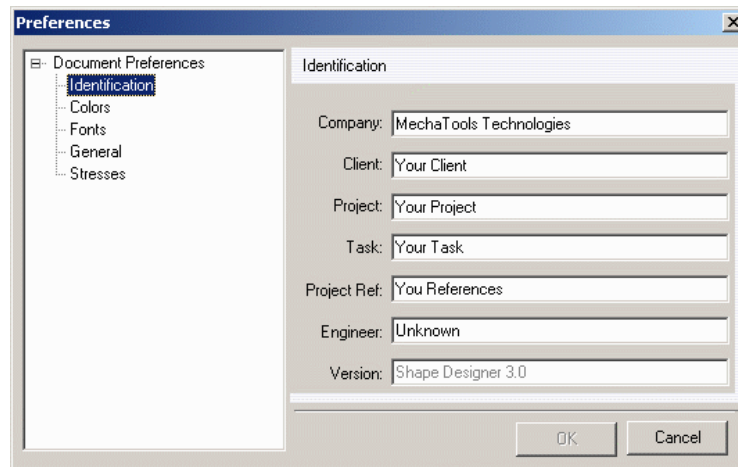
The **General** tab is used to set various general-purpose properties:

a) Number of real divisions per circle
b) Relative thickness beam for visualization
c) Loads draw width for visualization
d) Deformation (solution) draw width for visualization
e) The visual amplitude of the responses in effort (moment, shear, axial effort)
f) Grids spacing ticks
g) The *ShapeDesigner* software path (in option) for calculating complex geometric properties of sections
h) The *Twprofile* Path (in option) for calculating complex geometric properties of thin-walled open sections
i) Output visual solution refinement

The number of divisions per circle is used to specify the number of linear segments to approximate circles.
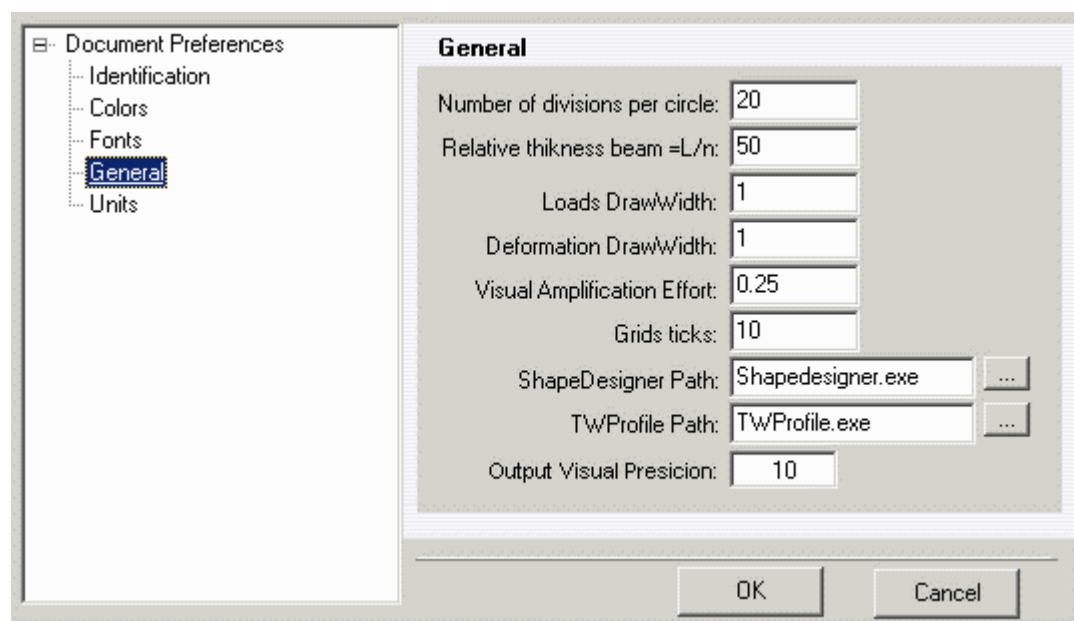
Creating Beams

From the **Toolbox** under the **Geometry Input** tab, click the beam icon. Then in the drawing area you can connect two nodes to create beam.
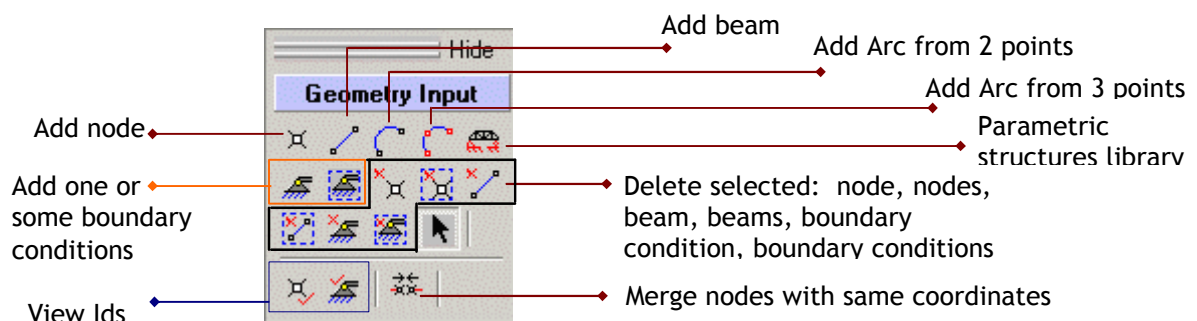
1) **Add Beam** by selecting two exiting nodes.

2) **Add Arc from 2 points** is used to create arc from 2 selected nodes and specified radius
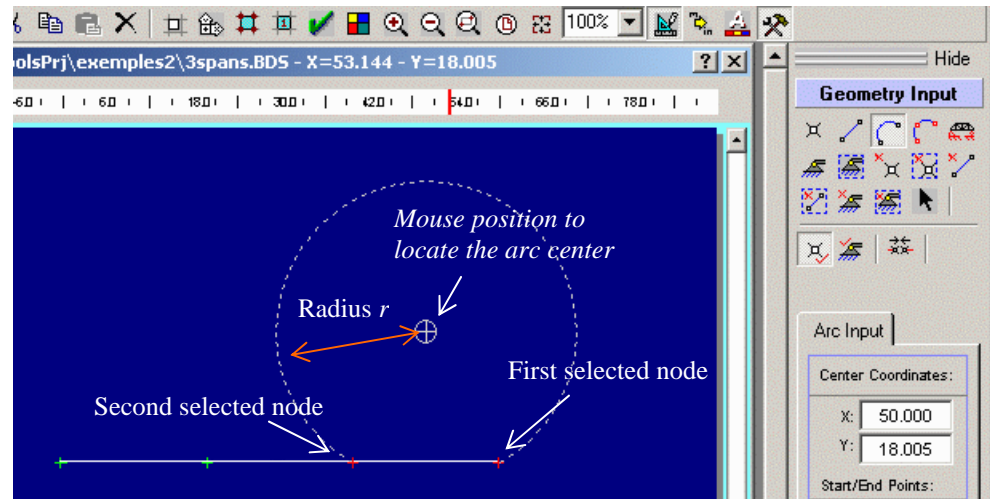
3) **Add Arc form 3 points** is used to create arc from 3 selected nodes.
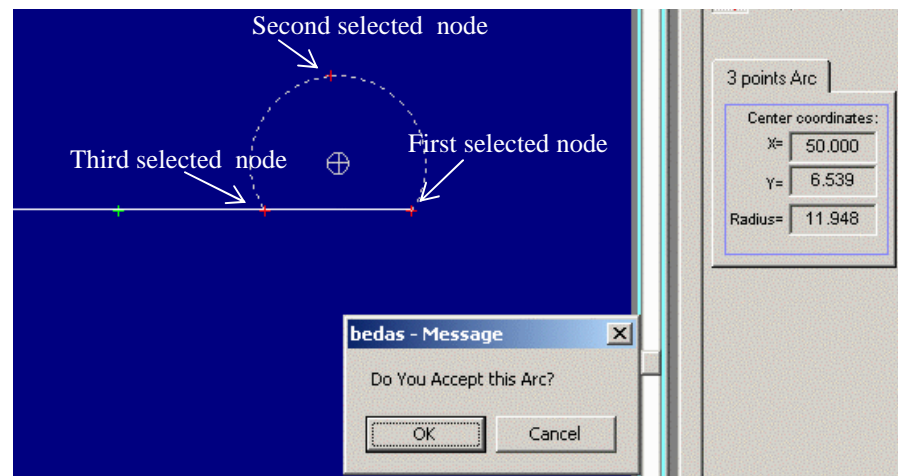
4) **Parametric Structures Library** is used to create parametric structures from the library.

Library browser

Structure view

Parametric dimensions

Confirm to accept selected structure

Accept selected structure
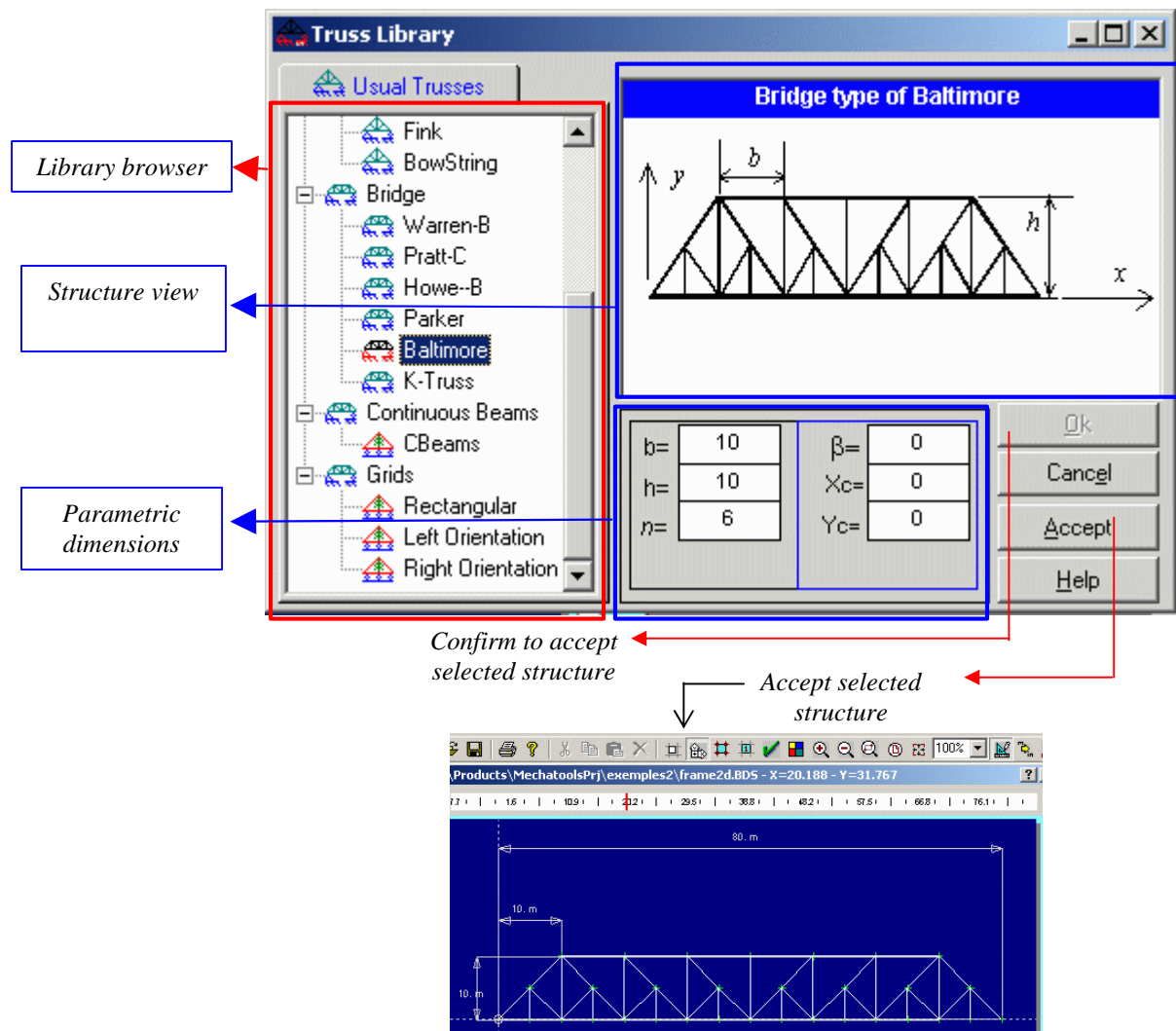
The method used to create beams and the number of beams required is specific to the problem to be solved. The following guidelines should be followed.

1) BEDAS assigns material properties to beams.
2) BEDAS automatically merge together common nodes. To detect common nodes boundaries, the distance between two boundaries is compared to a very small tolerance value (10^{-5}). If this distance is smaller than the tolerance value, the boundaries assumed to be merged. Users should be very precise when assigning point coordinates.

Creating Custom structures from data files

Use this option to input a beam geometry using a text data file. The text file format must follow the rules below:

1) The first line contains the number of nodes (n) followed by the number of beams (m).
2) Each of the following n lines contains the x and y coordinates of a polygon point separated by a comma.
3) Each of the following m lines contains the nodes beam connectivity i and j.

The following data file content describes a two beams structure:

3, 2   *// 3 nodes and 2beams*
0.0, 0,0 *// x1, y1*
1.0, 0.0 *// x2, y2*
3.0, 0.0 *// x3, y3*
1 2   *// beam 1 from node 1 to node 2*
2 3   *// beam 2 from node 2 to node 3*

Selecting beams

There are three ways to select beams.
1) Using the pointer
2) Using the **Beams browser**
3) Sing **Selection** icon

To select a beam using the pointer, move the pointer over the beam in the drawing area then press the left mouse button. The selected beam is shown in red. You can select more than one beam if you hold the **Shift** key down at the same time as you select beams by mouse.

To select a beam using the **Beams browser**, select the **CAD and Data** tab from the toolbox and select the desired beam directly from the list using the **mouse pointer**. The selected beam is shown in red.

To select multiple beams using a selection box, select the **CAD and Data** tab from the toolbox and click on the **Selection** icon. Drag the pointer, using the left mouse button, to draw a selection box around the beams you want to select. When you release the mouse button, all beams totally inside the selection box are selected. The selected beams are shown in red.

Deleting Beams

To delete beams, select the beams then press the **Delete** key or click the **Delete Beam** icon of the toolbar or from the **Edit** menu click **Delete**. A confirmation is always required to delete a beam.

Moving Beams with the Pointer (or arrow keys)

To move beams with the pointer, select the beams, then, from the **CAD and Data** tab of the toolbox, click **Move Selected Beams by Mouse** or from the **Beam** menu click **Move**. The selected beams become movable. Drag the beams using the left mouse button in the drawing area or using the arrow keys. When you release the left mouse button, the beams return to the unmovable state. If you want to move all beams, from the **CAD and Data** tab of the toolbox, click **Move All Beams by Mouse**.

See the Moving Beams Accurately structure of this document for a more accurate method.

Modifying Beams with the Pointer (Moving Nodes)

To move a beam node, select the beam you want to modify, then, from the **CAD and Data** tab, click **Modify Beam by Mouse** or from the **Beam** menu select **Modify Beam**. The beam becomes modifiable. The beam nodes will be highlighted. Drag the beam nodes using the left mouse button in the drawing area. When you are done modifying the beam, click the right mouse button to return to non-modifiable state. See the Modifying Beams Accurately structure of this document for a more accurate method.

Zooming

Zooming allows you to increase or decrease the size of the drawing viewport. There are four ways of zooming.

1) Using **Zoom In**/**Zoom Out** icons of the toolbar or the **View** menu **Zoom In**/**Zoom Out** commands, you may increase or decrease the viewport by 10%.
2) Using the zoom percentage dropdown you may increase or decrease the viewport by an arbitrary value.
3) Using the **Zoom All** icon of the toolbar, you may readjust the viewport so that the bounding box is fully displayed and centered on the drawing area. See the Setting the Bounding Box structure in this document.
4) Using the **Zoom Window** icon of the toolbar or the **View** menu **Zoom Window** command you may define a selection box with the pointer. Drag the pointer using the left mouse button to draw a selection box around the region you want to zoom. When you release the mouse button, the drawing area will be redrawn so that the selected region occupies the whole drawing area.

Using the Grid

To help position the pointer on the drawing area, it is possible to display/hide the grid. To toggle the grid display, from the toolbar click the **Show and Hide Grid** icon or from the **View** menu click **Grid**.

Copying Beams

There are two ways of copying beams: Using the **Copy** and **Paste** commands and using the **Generate Beam** dialog. See the Generate Beams structure of this document for description of the **Generate Beam** dialog.

Select the beams you would like to copy. From the **Edit** menu click **Copy** or press CTRL + C. From the **Edit** menu click **Paste** or press CTRL + V. The operations actually performed by this sequence are:

Rotating Beams

To rotate a beam, select it and from the **CAD and Data** tab of the toolbox, click on the **Transform beam** icon or from the **Beam** menu click **Transform**. The **Beam Transformation** dialog is displayed. Click **Rotate** to display the rotation options.
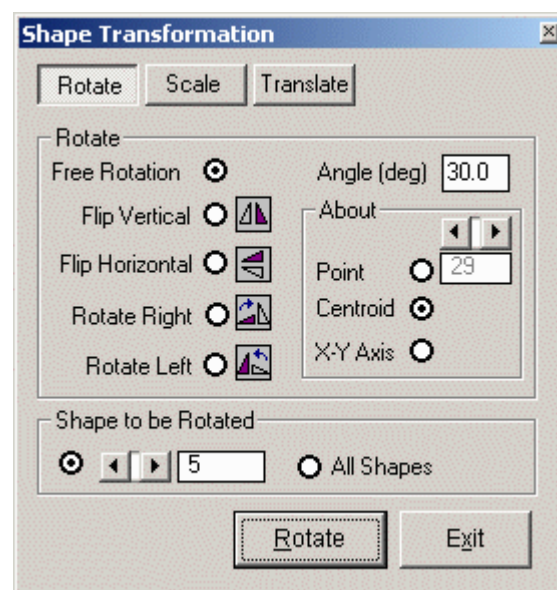
Select the beam you want to rotate in the **Beam to be Rotated** structure. You may select a specific beam or **All Beams**. Note that the horizontal scrollbar in the **Beam to be Rotated** structure serves two purposes. If specific beam is selected, it is used to select the beam to be rotated. If **All Beams** is selected, it is used to select the **About Point**. Select the rotation type using one of the five options available:

1) **Free Rotation** rotates the beam the specified **Angle**.
2) **Flip Vertical** flips the beam around a vertical axis.
3) **Flip Horizontal** flips the beam around a horizontal axis.
4) **Rotate Right** rotates the beam clockwise 90 degrees.
5) **Rotate Left** rotates the beam counterclockwise 90 degrees.

In the case of rotation transformations, **About** select the rotation point. In the case of flipping transformations, **About** fixes the position of the flipping axis. **About** may be one of the following options:

1) **Point** selects a specific geometric point of the selected beam in the **Beam to be Rotated** structure (even if a **All Beams** is selected). The point is highlighted in the view area.
2) **Centroid** uses the centroid of the selected beam or the centroid of all beams as the about point.
3) **X-Y Axis** uses the origin of the Cartesian coordinate system as the about point.

Click **Rotate** to apply the rotation.

The **Beam Transformation** dialog remains displayed for you to proceed to other transformations. Click **Exit** when you are done. Note that all transformations done in the **Beam Transformation** dialog are done in a single transaction and may be undone using the **Edit** menu and clicking **Undo**.

Moving Beams Accurately (Translate)

To move beams accurately, select a beam and from the **CAD and Data** tab of the toolbox, click on the **Transform beam** icon or from the **Beam** menu click the **Transform** icon. The **Beam Transformation** dialog is displayed. Select the **Translate** tab.
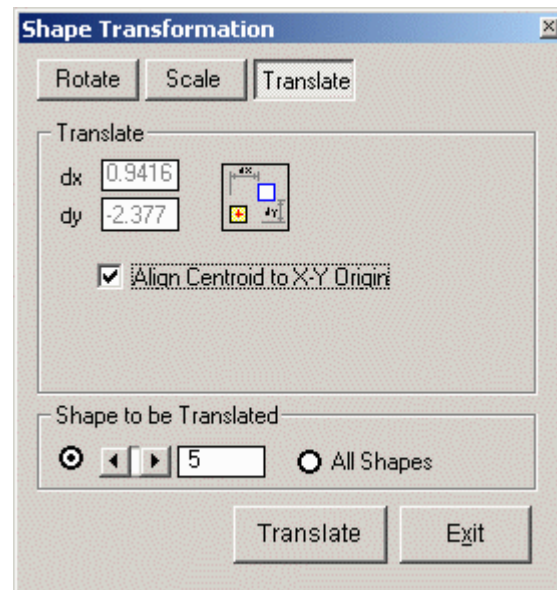
Select the beam you want to translate in the **Beam to be Transformed** structure. You may select a specific beam or all beams. Enter the translation values in the **dx** and **dy** textboxes. To align the centroid of the selected beams on the origin of the coordinate system, check the **Align Centroid to X-Y Center** checkbox. Click **Translate** to apply the translation. The **Beam Transformation** dialog remains displayed for you to proceed to other transformations. Click **Exit** when you are done. Note that all transformations done in the **Beam Transformation** dialog are done in a single transaction and may be undone using the **Edit** menu and clicking **Undo**.

The Beam menu offers the **Align Beam Centroid to X-Y Origin** and the **Align All Beam Centroid to X-Y Origin** shortcuts.

Modifying Beams (Moving nodes) Accurately
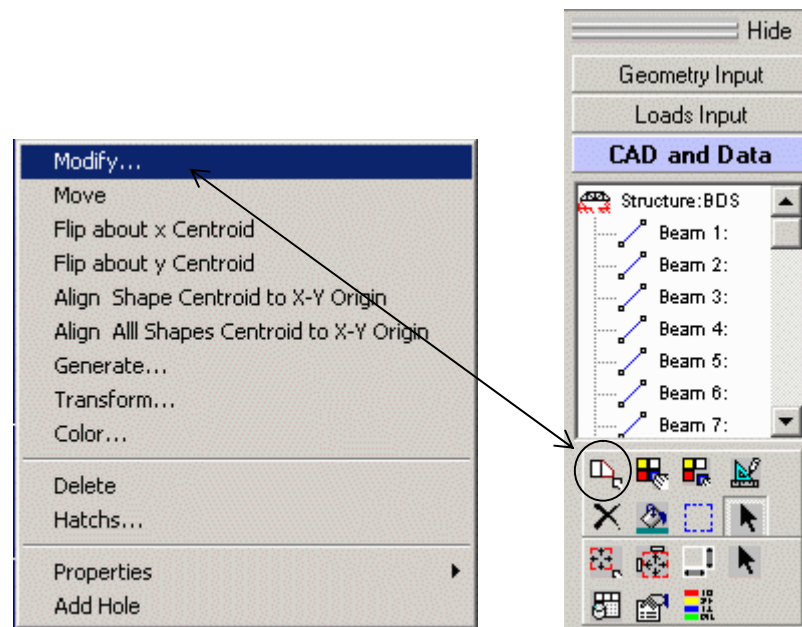
To modify the nodal coordinates of a beam, select a beam and from the **CAD and Data** tab of the toolbox, click the **Geometric Properties** icon or from the **Beam | Properties** menu click **Geometry**. The **Geometric Properties** dialog is displayed. Click the **Nodes** tab. You may modify the displayed nodal coordinate. The **Geometric Properties** dialog may be left open while working. Its content is updated dynamically.

Refreshing the Drawing Area

To refresh the drawing area, click on the **Refresh the Drawing Area** icon in the toolbar.

Refresh icon
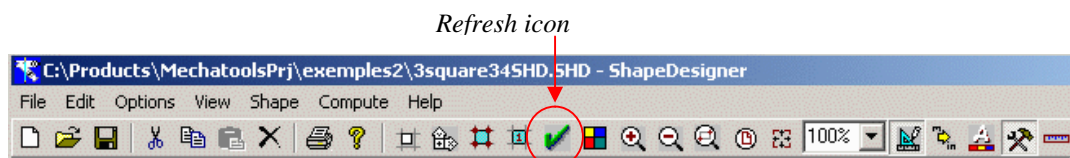
Setting Material Colors

It is possible to assign colors to materials. To do this, select the beams, and then from the **CAD and Data** tab click **Set Beam Color** or from the **Beam** menu click **Colors**. The **Color** dialog is displayed. Select a color and click **OK**. The color is assigned to the beam material. All beams using the same material will be affected by this change. To view beam colors, from the toolbar click the **Show and Hide Beam Colors** icon.

Setting Material Properties

To assign material properties to beams, from the **CAD and Data** tab of the toolbox, click **Beam Material properties.** You may also click **Material** from the **Beam | Properties** menu. The **Material Properties** dialog is displayed. Select the **Beam Materials** tab.
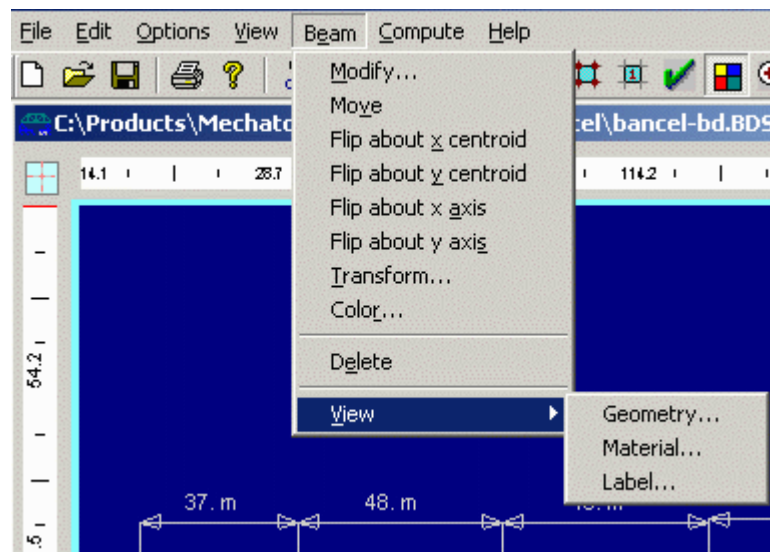
Select the beam to which you want to assign a material using the **Beam Number** horizontal scrollbar. For convenience, the beam **Label** is displayed and may be modified. Select the material type using the **Material Type** selection dropdown. The material properties are displayed. If your material type is not listed, select the *User-Defined* material type. When this option is selected, you may enter user-defined values for material properties. For user-defined material types, enter the following material properties: *Young's Modulus, Poisson Ratio and Mass Density*. If you want to use the same material type for other beams, check the beams in the **Select for Same Material** list. Click **Apply** to commit the current changes and keep the dialog open. Click **OK** to commit the current changes and close the dialog. Click **Cancel** to abort changes made since the last **Apply** and close the dialog.

In BEDAS, it is included two components to compute the geometric properties of the beam cross-section, the *standard steel section* (library of the AISC) and the *general parametric section*. You can also using the ShapeDesigner software (in option) as a plug in with BEDAS for geometric properties of the complexes composite cross-sections.

Bedas Materials Input and General Properties

For special cross-section click New section… .

Predefine and user-defined material list, includes the yield stresses

We can also input directly the section properties.

If we select section from the Predefined Sections tab, the section geometry is saved permanently

Standard Steel sections

General parametric Steel sections

At anytime you may select the **Material Properties** tab to get an overall view of material properties.

If you do not explicitly define the material type, it defaults to the first material in the material database.

Adding Labels to Beams (Notes)

To add labels to beams, select the beam to which you want to add a label. From the **Beam** menu, click on **view | Label…**. The **Set Beam Label** dialog appears. Enter a textual label then click **OK**. Anytime you want to view labels, from the **Dimensions** tab click the **View Beams Label** icon or from the **View** menu click **View Beams Labels**.

Dimensioning Beams

To view dimensions, from the **Dimensions and Labels** tab of the toolbox, click the **View Dimensions** icon or from the **View** menu and click **Dimensions**.

To dimension beams, from the **Dimensions & Labels** tab of the toolbox, click one of the three dimension types icons: **Horizontal**, **Vertical** and **Oblique**. BEDAS switches to *Dimension Definition Mode*. Move the cursor over the first vertex point to dimension. When the point is selectable, a small circle appears around the point. If the pointer is correctly positioned on the point, the pointer changes to an arrow beam. Click on the left mouse button. Repeat the previous steps for the second vertex point. When the second vertex is selected, the dimension is displayed and it follows the cursor. Position the dimension with the mouse and click the left mouse button to lock it in place.

When you are done dimensioning, click the right mouse button to switch out of *Dimension Definition Mode*.

Editing Beam Dimensions

To edit beam dimensions, click on the **Edit Dimensions** icon of the **Dimensions & Labels** tab of the toolbox. BEDAS switches to *Dimension Editing Mode*.

To move a dimension, press the **Move Dimension** button and select the dimension from the drawing area by left mouse button. Move dimension to the desired position. Click on the left mouse button to lock the dimension in place. Click on the right mouse button to cancel the operation.

In the toobox, select the **Dimension**s tab and from the *dimensions browser* select desired dimension then press the right mouse button to get the popup menu

Move Dimension button

Delete a dimension

Change the orientation of a dimension

Delete all dimensions

Set dimension label
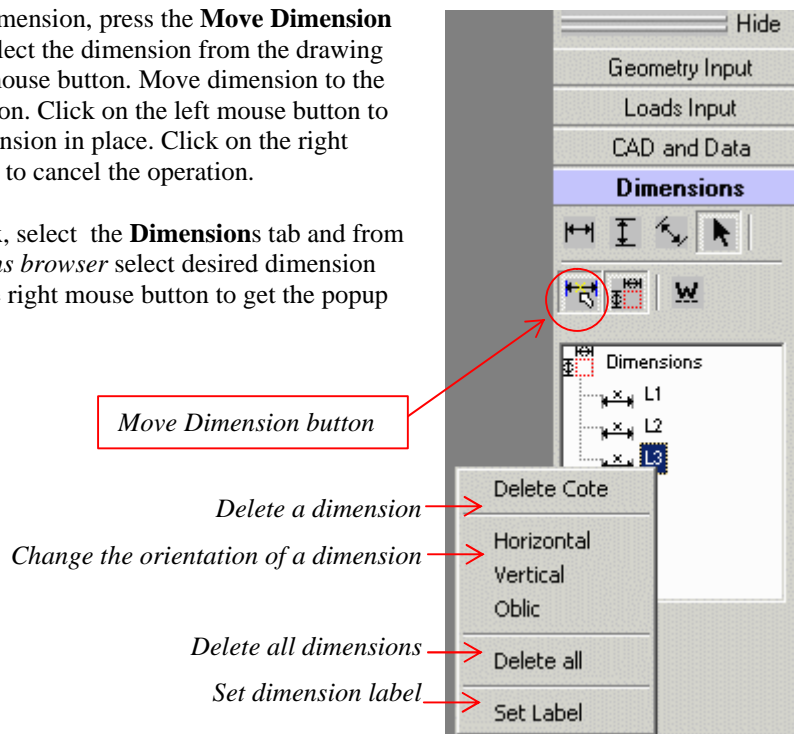
Saving a Project File

To save a drawing structure to a file, on the toolbar click the **Save File** icon or from the **File** menu click **Save**. The default file extension for BEDAS is "BDS". If this is the first time to save your work, the **File** menu **Save As…** dialog will be displayed.

Exporting Beams Structure to Other Applications

BEDAS exports files using the DXF format. This is the most popular CAD file format. To save a structure file in a DXF format, From the **File** menu click **Export | to DXF**. The current structure project filename suffixed with a DXF extension will be used as the exported file filename.

BEDAS can also save a snapshot of the drawing area to a bitmap using the **File** menu **Export | to BMP**. The current structure project filename suffixed with a bmp extension will be used as the exported file filename.

Computing dynamic analysis and/or mode shapes

To compute dynamic analysis and/or mode shapes, from **Compute** menu click **Compute** or press **F5.**
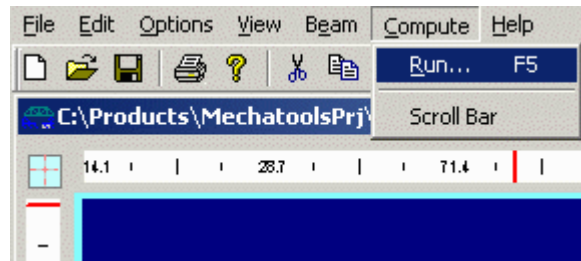
The **Dynamic analysis** dialog is displayed. This dialog is used to specify the type of analysis, the frequency max or the number of modes, the time step and the damping ration for dynamic analysis. There are two types of analysis:
1) Mode shapes and vibration frequencies computing
2) Dynamic analysis.

In Free vibrations computational and dynamics analysis you must input, the high frequency includes in the analysis or the maximum number of modes. To switch from the maximum frequency to the number of modes, clear the maximum frequency value and input the number of modes. Do the same to switch for number of modes to maximum frequency.

BEDAS - Dynamic Analysis

Dynamic Analysis

☑ Free Vibrations

Frequency Max: `0`   Number of Modes `10`

☑ Dynamic Response

Duration Time(s) `3`

Time Step `0.01`

Damping Type

$\alpha M + \beta K$ αM βK
○   ●   ○

mode `1`   ξ1 `0.03`
mode `0`   ξ2 `0`
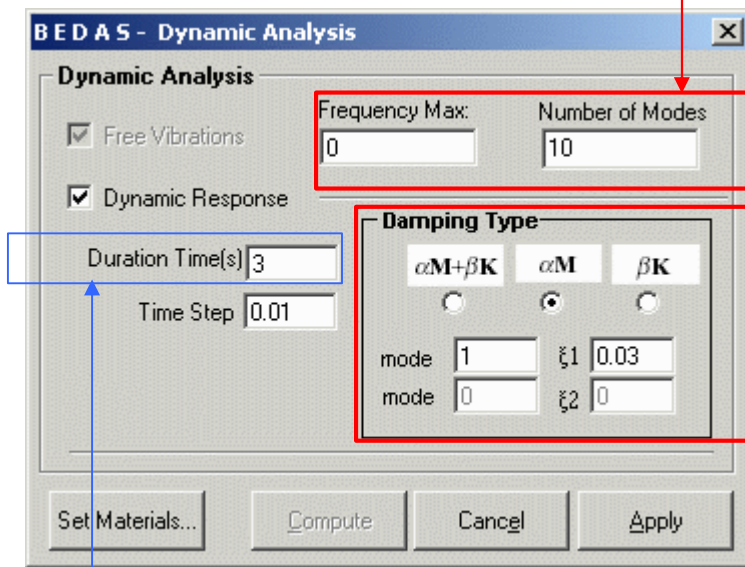
For dynamics analysis you can input the damping ratio if it's present in your structure. Damping coefficient is a percentage of the critical damping of the structure materials. Damping is considered a proportional viscous damping.

$$x_i = \frac{a}{2w_i} + \frac{bw_i}{2}$$

Set Materials...   Compute   Cancel   Apply

Total time (s) of the dynamic analysis

When computations are done, BEDAS switches to **Result** tab of toolbox. In this mode, you can view mode shapes animation and dynamic responses

Printing

Choose Print from the File menu

Chapter 4

Commands and Toolbars

Users of BEDAS may use the icons on the toolbars to speed up access to some commonly used functions. Holding the cursor over an icon displays a tool tip describing the icon function.

Main Toolbar (main toolbar is rotate to -90 degrees)

New File

Open File

Save Current File

Print graphical results

Help

Cut selected beams

Copy selected beams

Paste selected beams

Delete selected beams

Show and hide gridline

Show and hide axes

Show and hide points Ids

Show and hide beams Ids

Refresh graphics

Show and hide materials colors

Zoom in

Zoom out

Zoom window

Zoom All

Set automatically bounding box to fit all beams

Zoom list

Show and hide ruler

Change units

Set preferences

Show and hide toolbox

Results window

Toolbox

The toolbox is displayed by clicking **Toolbox** in the **View** menu. The toolbox may be moved from right to left of the application window by double clicking its title bar or press the right mouse button. The Toolbox contains icons that perform various operations. These icons are grouped by topics under tabs. For example, the **Geometry Input** tab contains icons used to manipulate geometries. Some of the toolbox icons do not have corresponding menu items.

Geometry Input

The **Geometry Input** tab contains operations to create nodes, beams, arcs, circles, and boundary conditions and to delete nodes, beams, and boundary conditions. Also to view nodes, boundary conditions and merges nodes.
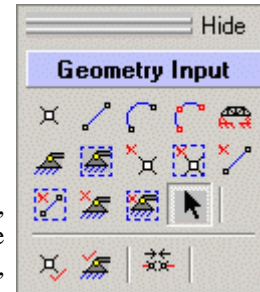
Loads input

The **Loads input** tab is used to add or modify loads and there properties and to verify the input load data.
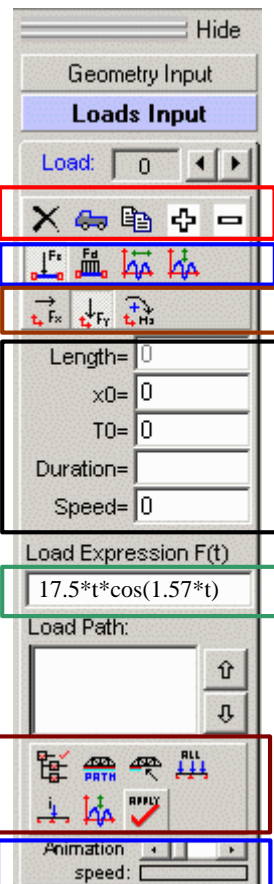
Delete all loads, add standard vehicle, duplicate current load, add empty load and remove current load

Load type: Concentrate, distribute, horizontal seismic and vertical seismic

Load direction in local beam reference: horizontal, vertical and bending

1. *Length* *=width of distribute load*

5. *Speed* *=v*

F_i

2. x_0 *Start position*

Path of current moving load

$t=0$

3. T_0 — *Start time of the current load*

4. *Duration*

Tf = *total time of the analysis*

time duration of the current load
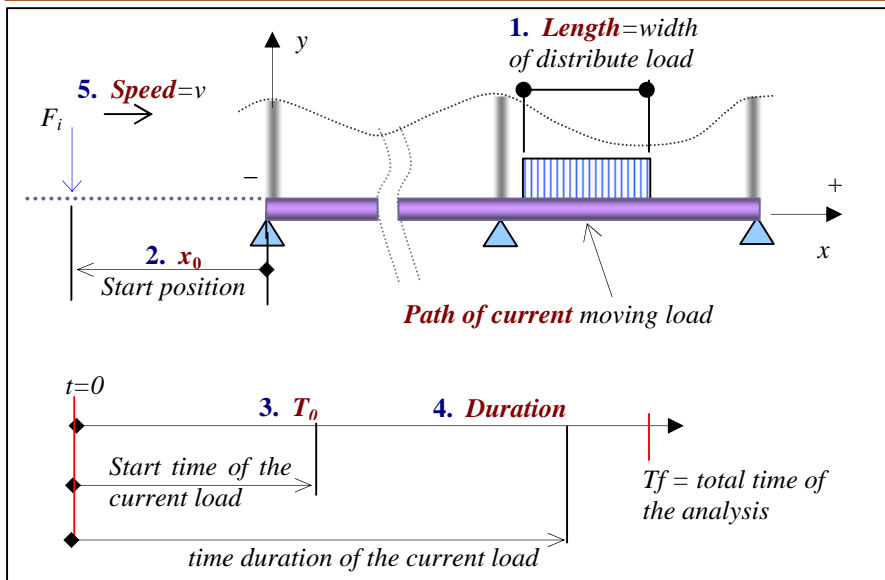
*Current load expression in numerical value or mathematical expression, for load variation given in file (ex. seismic) type: "**File:**" to browse the filename*

Set load path from list box, view load path, set load path by mouse, view all loads, view current load, display graphics load variation, and apply changes

Decrease or increase the animation speed of the visual simulation moving loads

Case of moving loads

The sort in path beam list is import because the load move from the first selected to the end one.

v

| Beam number | 7 | 6 | 5 | 4 | 3 | 2 |

| Path beam number | 1 | 2 | 3 | 4 |

Path of current moving load

From the figure: the path beam list of the current load is < 6   5   4   3 >, and the sort is very import
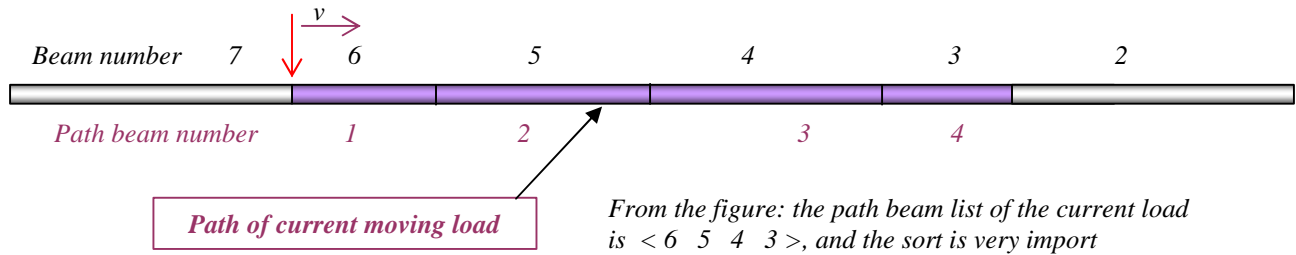
Case of moving loads – two loads in opposite directions

In this section, we show how to set the loads path and properties for two opposite directions (*see example in .../examples/3spans2.bds*)

F_1 v_1 v_2 F_2

| Beam number | 7 | 6 | 5 | 4 | 3 | 2 |

| Node number | 1 | 2 | 3 | 4 | 5 |
| Path beam number | 1 | 2 | 3 | 4 |

Path of the two moving load

From the figure:
1.The path beam list of the load F1 is < 6   5   4   3 >, the speed is v1and x0=0
2.The path beam list of the load F2 is < 3   4   5   6 >, the speed is –v2 (negative) and x0=l_3 (length of the first beam in the list (beam number 3).
x0 is the start position

Load Expression F(t)
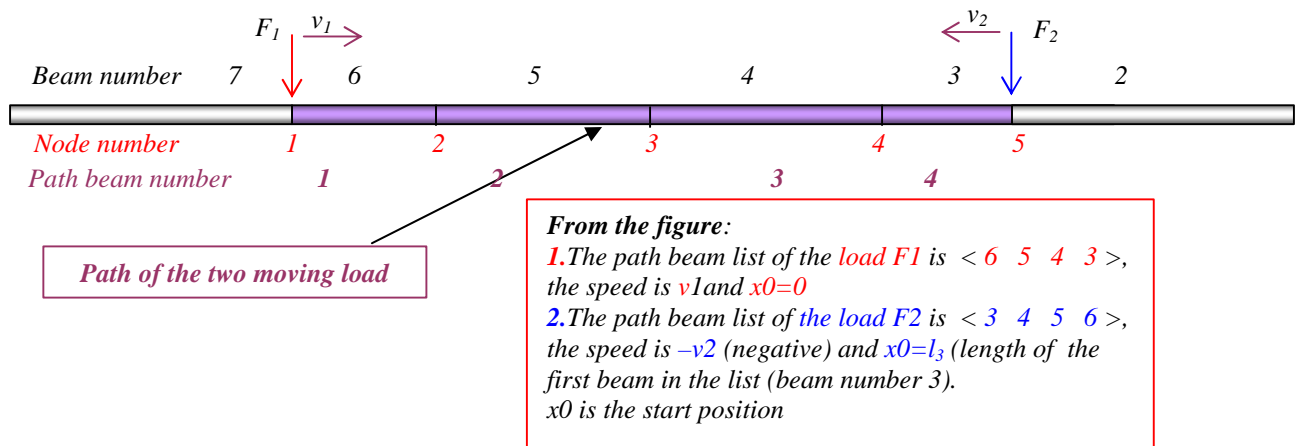
Load Path:

Use the up and down buttons in order to sort the path beam list in the desired and correct path

Animation speed:
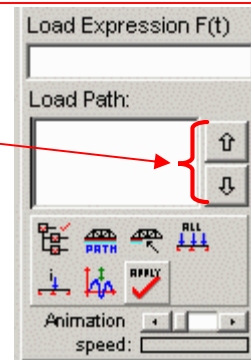
Case of fixed loads

In the case of fixed loads it is important to set *speed=0* and in the beam list of load path include just the beam where the load is applied.

x_0

| Beam number | 7 | 6 | 5 | 4 | 3 | 2 |

| Path beam number | 1 |

Path of current fixed load

From the figure: the path beam list of the current fixed load is <6> and the distance x_0 is computed from the origin of the beam

MechaTools Technologies Inc
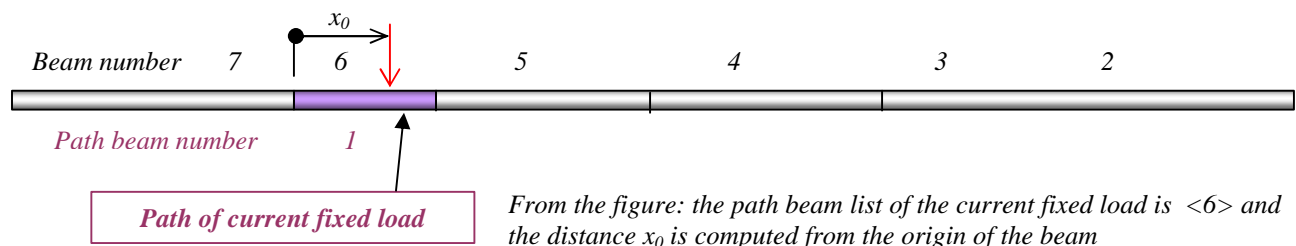
Case of fixed distributed loads

In the case of fixed distributed loads it is important to set **speed=0** and in *the path beam list* of current load include just the beams where the load is applied.

From the figure: the path beam list of the current fixed load is < 6 5> and the distance x_0 is computed from the origin of the first beam of the path beam list
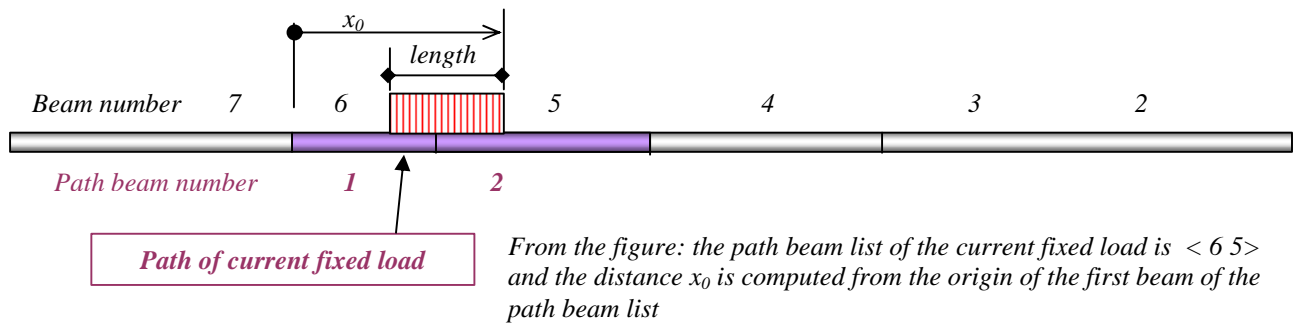
Standard Vehicles and trains

In standard vehicles and trains dialog, you can select specific standards loads (vehicles) easily and quickly. In standards loads, when you set parameters, like speed, duration, initial applied time etc., these parameters are applied to all loads of the selected vehicle and if you delete one axle (load) of the selected vehicle, Bedas asks you if you want delete all the vehicle axles. You can add to the database your user-defined vehicles if there are not present in the list. To add user-defined vehicles permanently you will add them in an ASCII file named "**Customveh.dat**". This file is present in you BEDAS folder.
The format of the content of this file is:

```
Number of vehicles           //do not change the three first lines there are for guideline
VehicleName, Number of axles/vehicle //
Force, LoadType, Idof, length, StartPosition, Tinitial, Duration1, Speed / for each axle //
2
Custom3axles, 3
-1000,"CONCENTRATE","V", 0,   0, 0, 10, 1
-2000,"CONCENTRATE","V", 0, -10, 0, 10, 1
-3000,"CONCENTRATE","V", 0, -20, 0, 10, 1
Custom2axles, 2
-4000,"CONCENTRATE","V", 0,0,0,20,2
-5000,"CONCENTRATE","V", 0, -20,0,20,2
```

First vehicle: Custom3axles

Second vehicle: Custom2axles
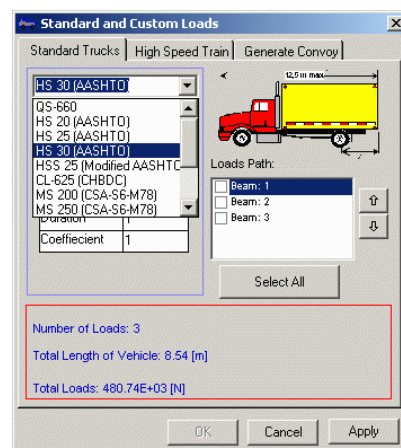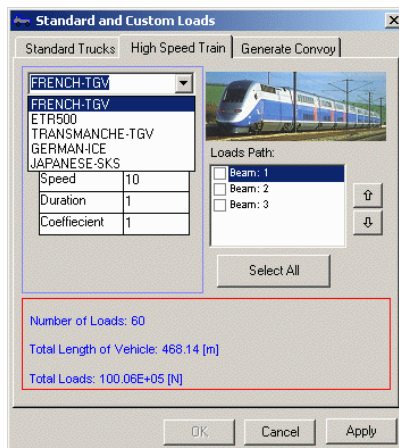
Seismic loads

In BEDAS, user can perform seismic analyses due to a specific accelerogram (e.g. El-Central etc.) Alternatively, under the effects of an acceleration of the ground motion, in the form of mathematical expression a(t). The seismic excitation can be vertical or/and horizontal.

To specify a filename, write: **File:** *and press return, Bedas gives you the possibility to browse your filename*
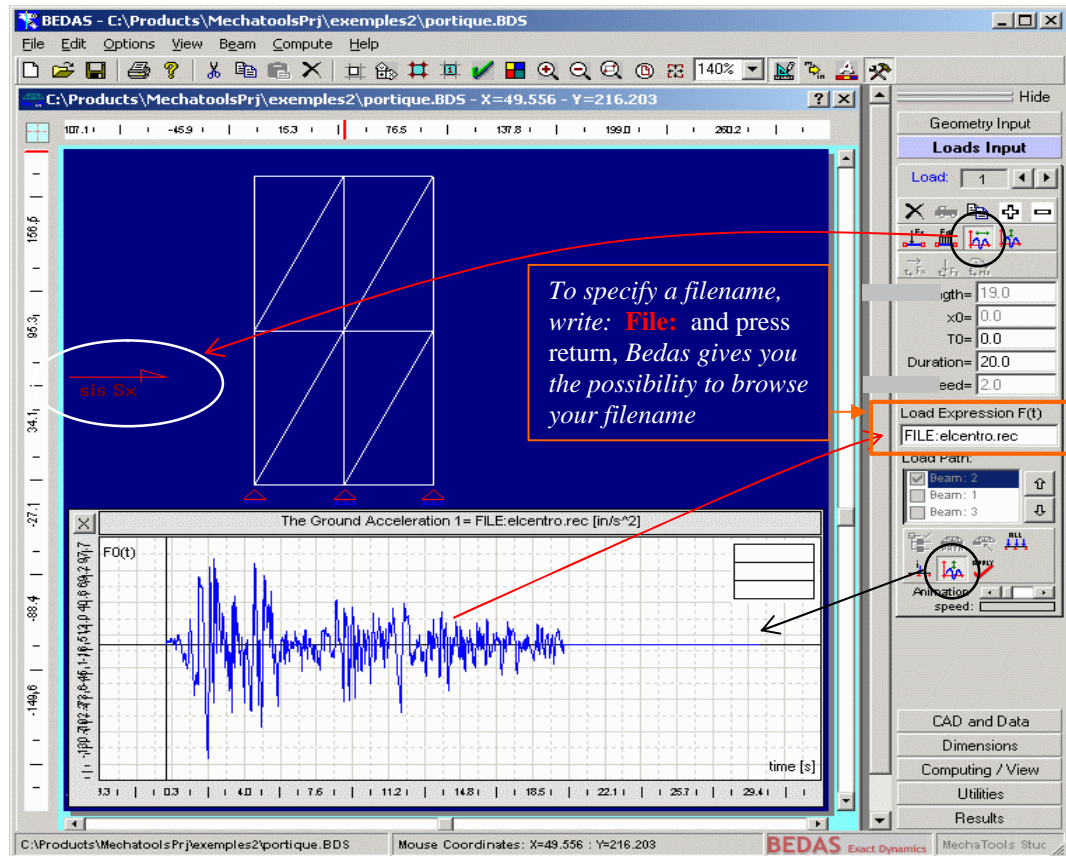
CAD and Data

The **CAD and Data** tab is divided in three categories. The first category contains operations to modify existing beams. The second category contains operations to change the bounding box. The last category is used display beam information windows.

Beams browser

Modify beam, move all beams, move selected beam, transform selected beam, delete selected beam, color selected beam, select beams by mouse

Set bounding box by mouse, set bounding box by values, show scrollbar, view beams coordinates table, show beams properties table, view beams material properties

MechaTools Technologies Inc
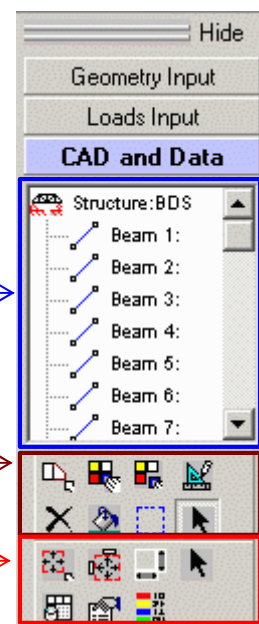
Computing
The **Computing** tab is used to execute and select the analysis type

Dimensions and Label
The **Dimensions and Label** tab contains operations to visually document the structure.

Results
The **Results** tab contains operations to visually view the mode shape, mode shapes animation, dynamic response and dynamic animation and more.
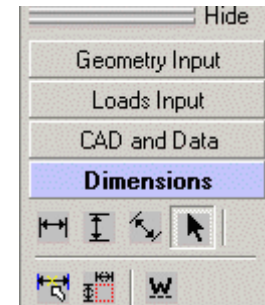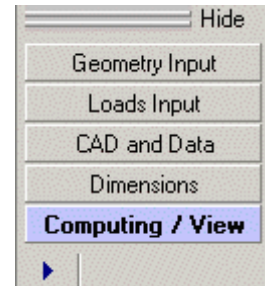
View beam mode shape info by mouse click, animate mode shape, and view tabulated results

*The Visual amplification response factor. You must **increase** this value if the visual solution **is not visible**, and you **decrease** this value if the visual solution **is exaggerate**. This factor may takes values from 0.0001 to 10000 and more. There is no influence on the real responses it is simply for drawing visual solution*
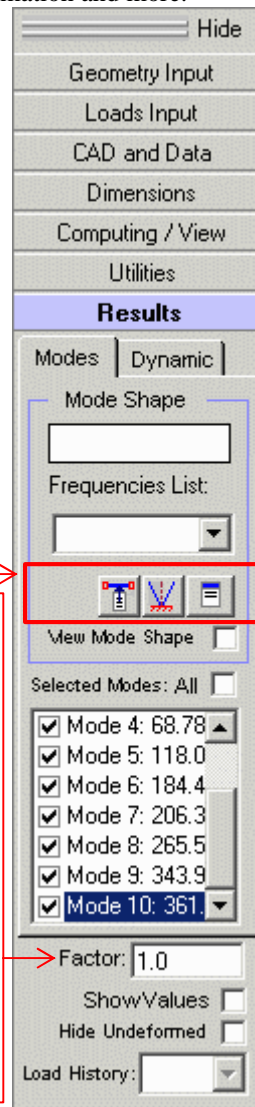
Static responses, Dynamic responses, and list of time steps.

View dynamic response in displacement, internal moment, shear effort and axial effort

View loads with response, decrease, increase the visual amplitude of the internal efforts, decrease, and increase the visual line width

Animation speed, view animate solution, view tabulated results.

View graphical time history of the numerical gage located at specific beam in local position x/L (L is the beam length).
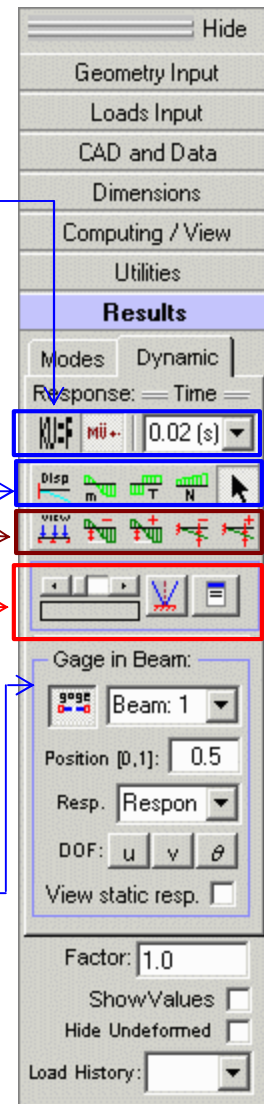
Boundary conditions

To set or modify the boundary conditions:

1. Press the boundary condition buttons. ⟶

2. Set the desired boundary conditions in the dialog

3. In drawing area, you select one node by clicking the mouse on the node or by selecting area that contains many nodes as in the structure
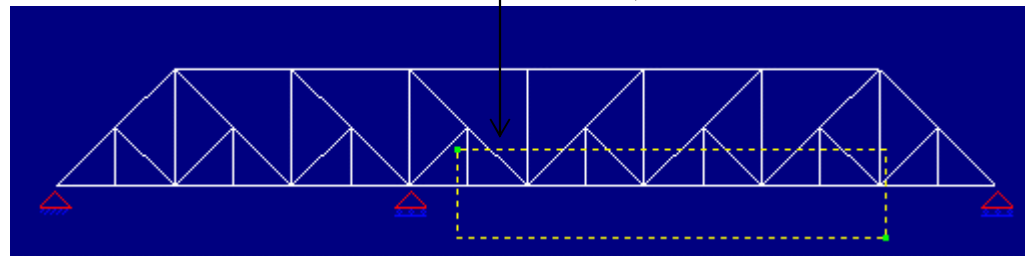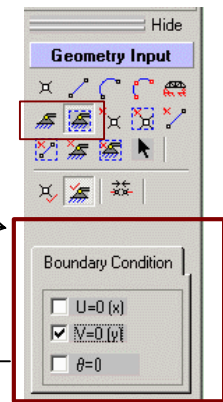
Boundary conditions applied to the selected area
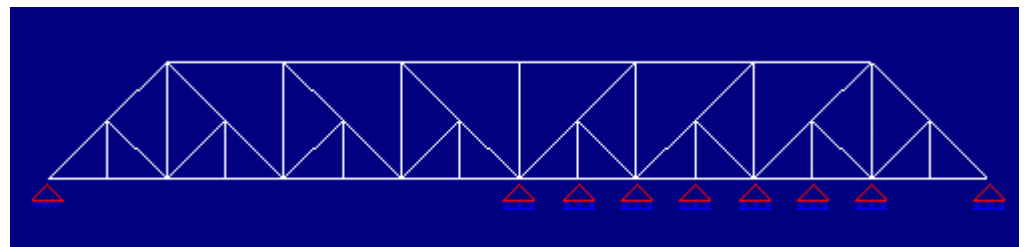
Menus

BEDAS uses standard Windows menu for **File**, **Edit** and Windows operations. It also has a range of menus for working with structures and groups of beams.

File Menu

The **File** menu contains commands for creating, opening and saving structure project files. It also contains commands to print and export data.
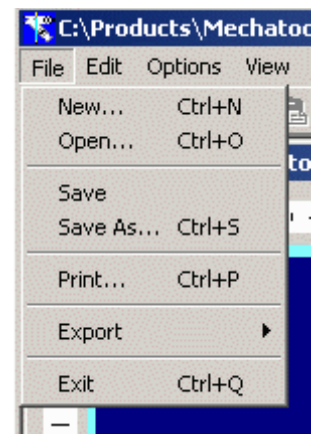
New (Ctrl + N)

Use **New** to start work on a new structure project. If you have unsaved changes to the current structure project, BEDAS will prompt you to save your work before starting the new project.

Open (Ctrl + O)

Use **Open** to open an existing structure project. If you have any unsaved changes to the current structure project, BEDAS will prompt you to save your work before opening the structure project.

Save (Ctrl + S)

Use **Save** to save the current structure project.

Save As

Use **Save As** to save the current structure project to a new file.

Print

Use **Print** to print the contents of the display area.

Export

Use **Export** to export the current structure project to a DXF file or to a bitmap file. The files are automatically created in the same directory as the structure project file (BDS). The file name used is the file name of the structure project except for the extensions (.BMP, .DXF). The bitmap file contains a view of the current drawing area. The DXF file contains the structure geometry.

Exit

Use **Exit** to exit the application.

Edit Menu

The Edit menu contains commands for copying and pasting tabular data, selecting objects and working in tables.
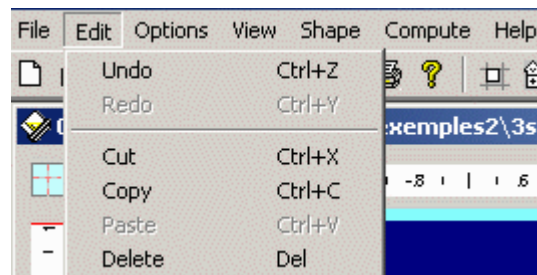
Undo

Undoes the last action carried out. **Undo** supports only one level of undo. When an action is undone, it may be redone using **Redo**.

Redo

This action redoes the last undo action. When an action is redone, it may be undone using **Undo**.

Cut (Ctrl X)

Remove the current selection and place it on the clipboard. (This info cannot be used outside the application)

Copy (Ctrl C)
Copy the current selection to the clipboard

Paste (Ctrl V)
Paste the contents of the clipboard into the current selection

Delete (Del)
Remove the current selection without placing it on the clipboard.

Options Menu
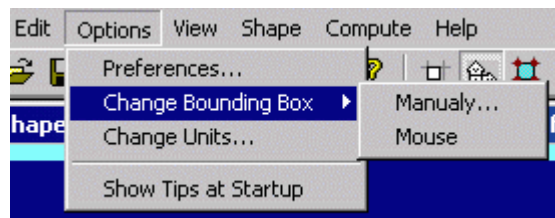
Preferences...
Change the options to values that you would like to use for most beams that you will create in the future (ex comment, project number, company, client, colors, fonts etc.).

Change Bounding Box

Mouse
Scale the drawing in the drawing area so that it just fits inside the window.

Manually
Set the maximum and minimum coordinates available in the drawing area. Use this to set up the overall coordinates before you begin drawing a structure.

Change Units...
Change the current units
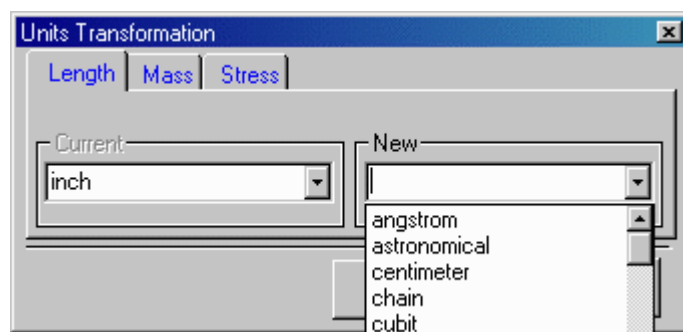
View Menu
The View menu contains commands for controlling the display in the drawing area.
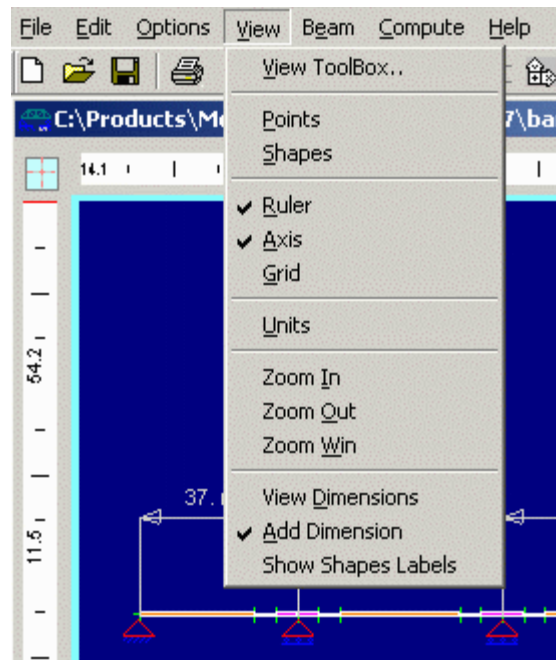
View Toolbox...

To see or hide the toolbox

Zoom

Zoom in on part of the current display. Press the mouse button and dragging a rectangle surrounding the area of interest. Release the button to draw the zoomed view.

Points

Turn on or off the display of points numbering of the beams in the drawing area

Beams

Turn on or off the display of Beams numbering in the drawing area

Ruler

Turn on or off the display of rulers in the drawing, area, double clicking on the ruler, set the spacing of the grid and ruler

Axes

Turn on or off the display of axes in the drawing area

Grid

Switch on or off the use of the grid in the drawing area and set the spacing of the grid.

Units...

Change the current units

View Dimensions

Turn on or off the display dimensions in the drawing area

Add Dimension

Turn on for adding dimensions annotation in the drawing area

Show Beams Labels

Turn on or off the display of Beams labels in the drawing area

Beam Menu

The Beam menu contains commands for working with Beams in the area
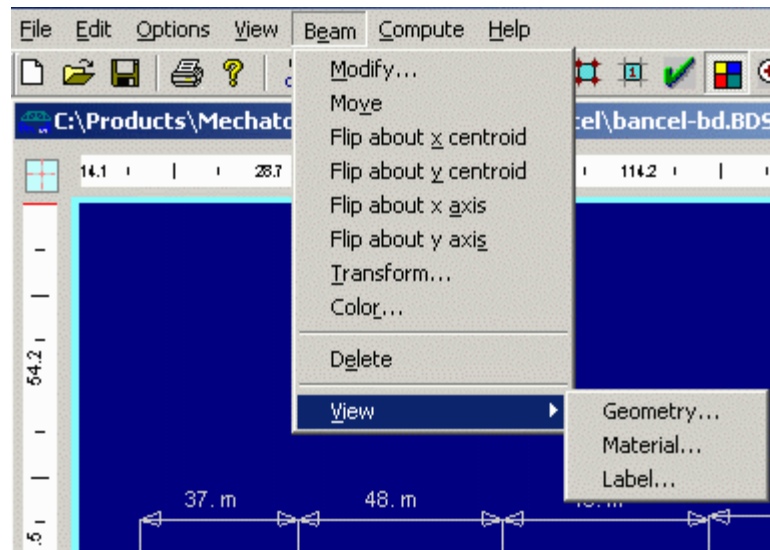
Flip Horizontal (Flip about X Centroid)

It reflects the selected beams about a vertical axis passing through origin or the centroid of the area of the selected beam.

Flip Vertical (Flip about Y Centroid)

It reflects the selected beams about a horizontal axis passing through the origin or the centroid of the area of the selected beams.

Transform beams
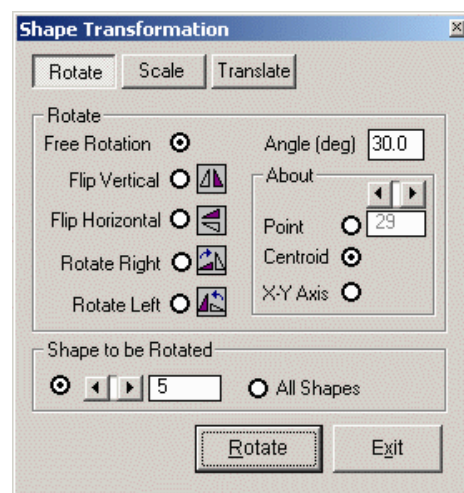
Duplicate

This command duplicates the selected beam in the work area window a given number of times. In this case, you can duplicate beam by rotation, flipping, scaling or translating.

Rotate

It rotates all or the selected beams in the drawing window a specified number of degrees about the origin of the axes, the centroid or a specific point in the selected beam. A dialog allows you to enter the number of degrees of rotation. Rotation is positive anti-clockwise.

Rotate Right

Rotates all or the selected beams in a 90 degrees about the origin of the axes or the centroid of each beam.

Rotate Left

Rotates all or the selected beams in a -90 degrees about the origin of the axes or the centroid of each beam.

Scale

It multiplies the coordinates of the selected beam by a specified scaling factor in each axis direction.

Translate

It allows you to move the selected beam a specified distance this provides a more accurate way of moving beams rather than dragging them with the mouse.

Get Length and Centroid

Display the table of selected beam properties.

Compute Menu
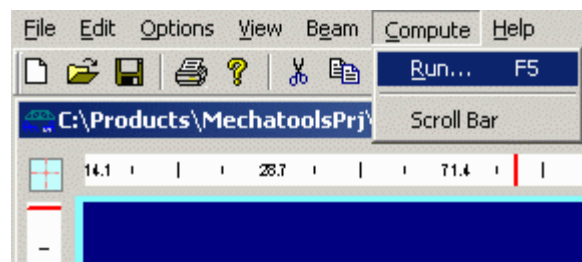
Run...

This command is to set materials, some parameters and execute properties calculations engine.

Help Menu

The Help menu contains commands for accessing BEDAS 's on-line he system.
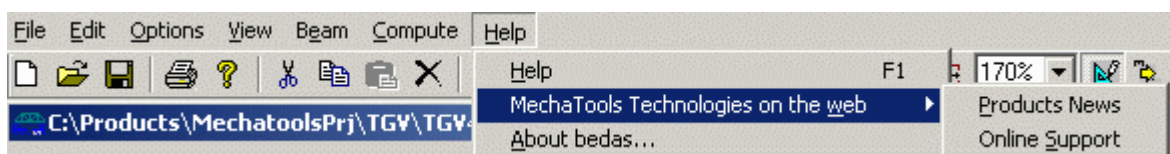
About BEDAS

It displays version information for this version of Beam Designer.

Selecting the Type of Analysis

The **Dynamic analysis** dialog is used to specify the type of analysis, the frequency max or the number of modes, the time step and the damping ration for dynamic analysis. There are two types of analysis:

1. Mode shapes computing
2. Dynamic analysis.

For more info see **chapter 3**, **Computing dynamic analysis and/or mode shapes** section
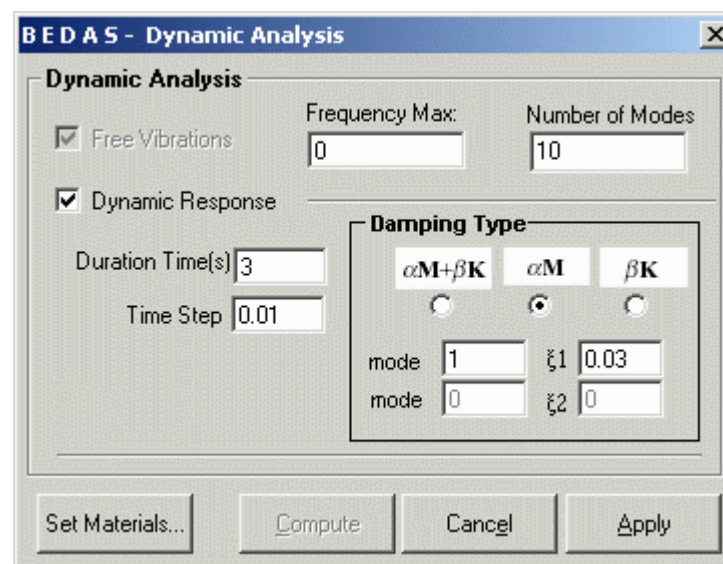
Computing

The **Apply** button validates all input and modifying data for the type of analysis before allowing computation. If the validation is ok, the **Compute** button becomes accessible. Click **Compute** to begin computations. Some time when you input a time step, BEDAS check the value automatically and suggests to you a coherent time step value. You can accept or refuse the suggested value. Some suggested value required a large number of steps (more than 5000 steps) so when you decide to accept the new time step check the number of steps in the warning dialog as fellow:
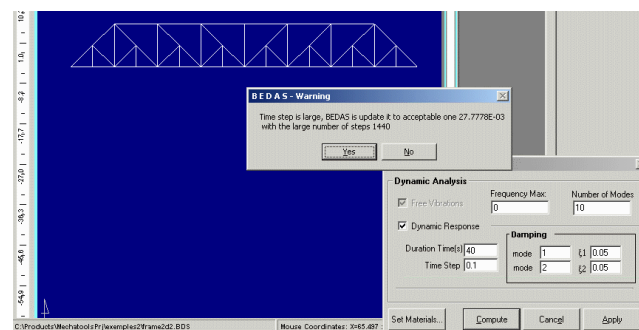
Result Mode

When computations are done, BEDAS switches to **Result** tab mode. In this mode, you can visualize and animate solutions. You can view mode shapes animation, dynamic responses distribution in displacement, bending moment, shear and axial internal forces and you can also plot time history responses in specific positions on the beams.

Mode Shapes Results
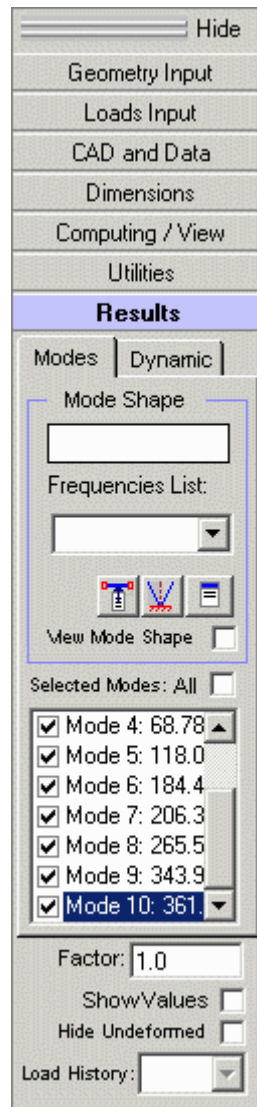
Beam mode shape info
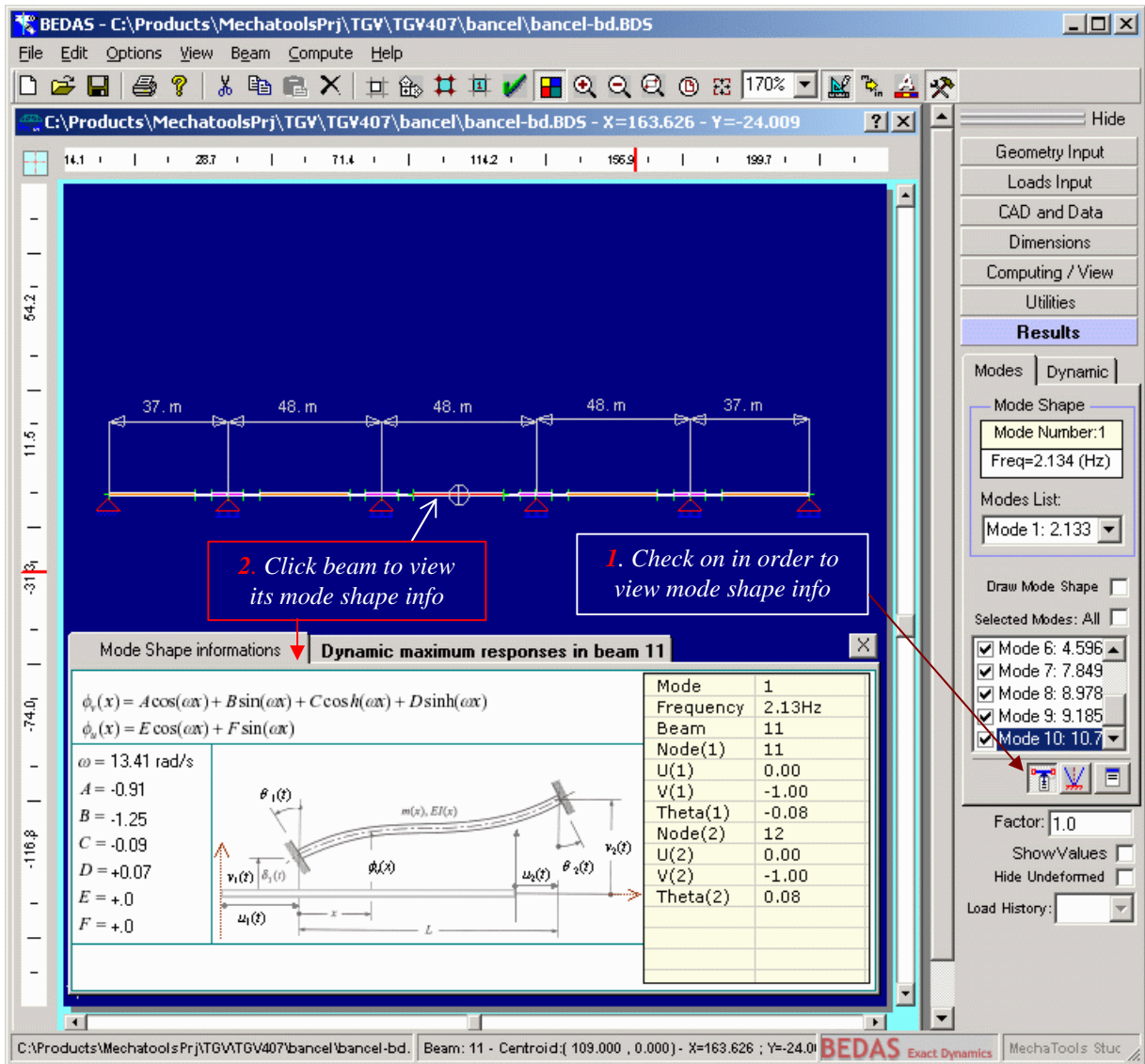
Mode shape info gives you the information about the analytical mode shape functions, the frequency, the nodal values of the current mode shape and the dynamic maximum responses in the selected beam by one click.

The Mode shapes functions are:

$$Bending: \boldsymbol{f}_v = A\cos(\boldsymbol{a}x) + B\sin(\boldsymbol{a}x) + C\cosh(\boldsymbol{a}x) + D\sinh(\boldsymbol{a}x), \boldsymbol{a}^4 = \frac{w^2 rA}{EI}$$

$$Axial: \boldsymbol{f}_u = E\cos(\boldsymbol{a}x) + F\sin(\boldsymbol{a}x), \boldsymbol{a}^2 = \frac{w^2 rA}{EA}$$

Dynamic responses results
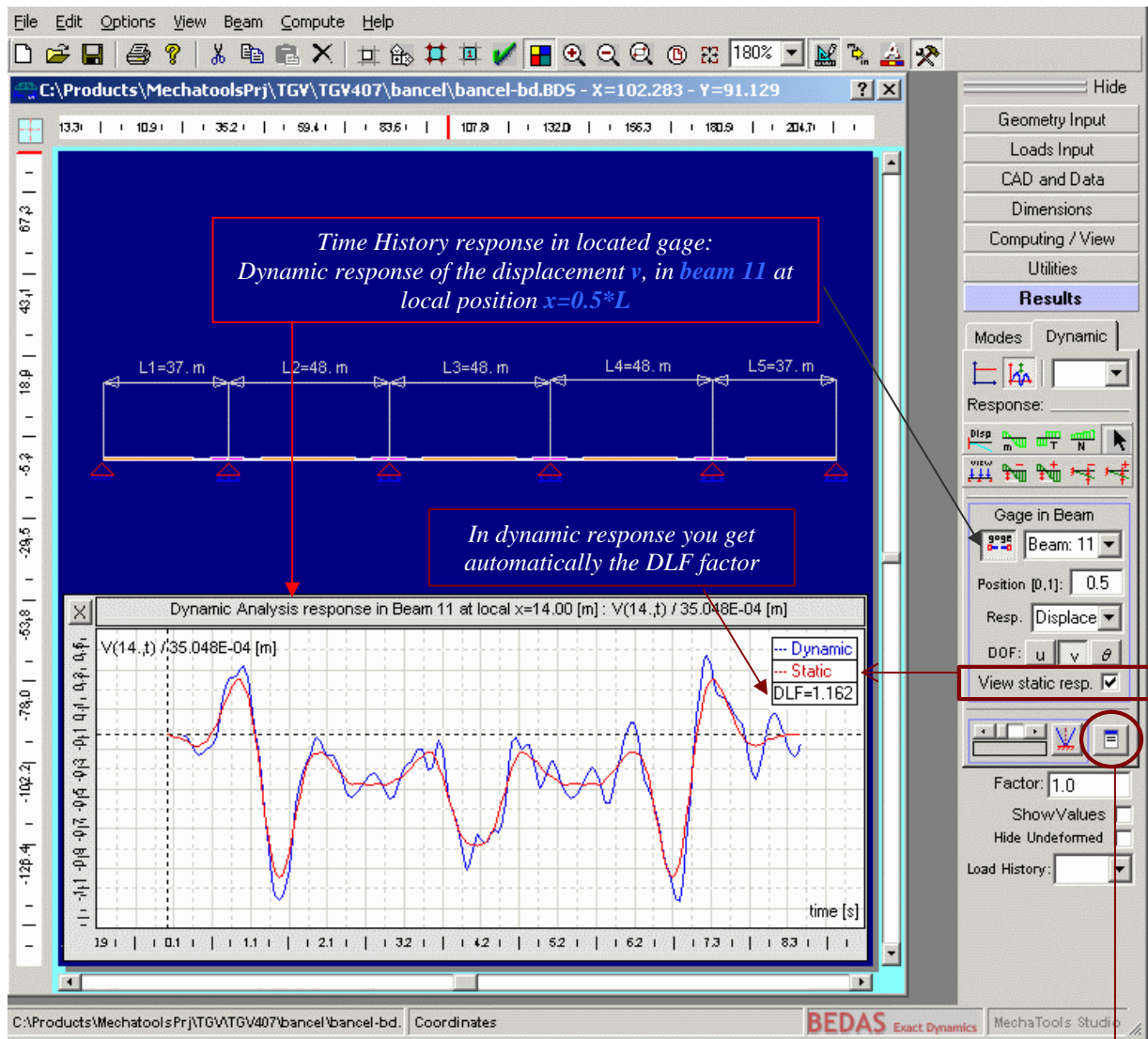
The relative gage position in beam [0,1]= x/L:
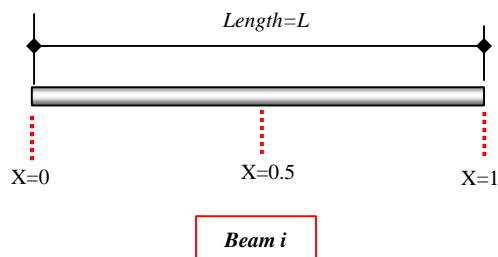
Viewing tabular dynamic responses results

The content of the **Results** dialog makes up the data that may be saved and/or printed.

Results		

BEDAS Resu...	Item	
Engineer:	You	
Project :	Your Project	
Comment :	Your Task	
Units :		
Length:	m	

Mode Shapes and maximum responses | Distributed Dynamic Responses | Time History Dynamic Responses

Mode number	Frequencies (...	Eigenvalues	Periods (s)	
Applied at position:	-46.8140E+01			
Path of moving loads is:	1, 2, 3, 4, 5, 6...			
-				
Dynamic Resolution Propertie...				
Time response duration:	85.0000E-01			
Time step response :	46.4481E-03			
Damping coeff. alfa,beta:	0.0, 0.0			
Damping ratio:	0.0, 0.0			
Global Maximum Dynamic R...				

Maximum Responses	Value	position [...	time [s]	Beam
Displacement U	+00.0000E+00	0.000	+00.0000E+00	0
Displacement V	-43.5416E-04	0.455	+72.4590E-01	16
Rotation Theta	-29.0999E-05	1.000	+72.4590E-01	14
Bending Moment	-62.3116E+05	1.000	+44.5902E-01	13
Shear Force	-73.1833E+04	1.000	+37.6230E-01	8
Axial Force	+00.0000E+00	0.000	+00.0000E+00	0
Velocity Vx	+00.0000E+00	0.000	+00.0000E+00	0
Velocity Vy	+39.8507E-03	0.364	+80.3552E-01	21
Velocity Vtheta	-36.3267E-04	1.000	+80.3552E-01	21
Acceleration Ax	+00.0000E+00	0.000	+00.0000E+00	0
Acceleration Ay	+93.9143E-02	0.636	+39.9454E-01	1
Acceleration ATheta	-85.6265E-03	0.000	+41.3388E-01	1

Equivalent visual results

Equivalent visual results
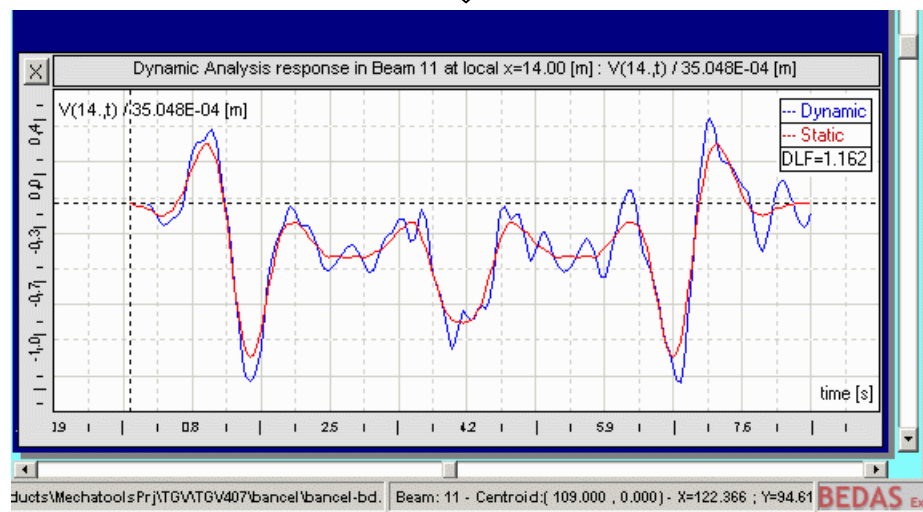
Design verification

The design of beams for specific applications is usually governed by detailed specifications and codes involving design requirements and procedures specific to each country.

In BEDAS, you can verify your design by introducing your custom code equations. Bedas accept the mathematical input formulas to verify the design of your structure by applying the specified equations in an collection of criteria. Each criterion contains a mathematical equation that respects one criteria of the code design. With this approach, BEDAS is a self-update design criteria code. If the country code changes, you can do it with the same BEDAS version without updating or purchasing a new version. From the **Results** dialog select the *Design verification* button, and the **Design verification** dialog appears

Current design Criteria number

The mathematical input design criteria: you must input left part equation, the operator and the right part equation. In criteria equation you must use only the variables contains in the parameters list

Parameters list (list of available variables that we can use in the design criteria equation)

Check all design

Add and remove design criteria

Results table

For example, for Dynamic amplification factor, we can input:
 vmax/vmaxst in the equation criteria first part and in the equation criteria last part we can input the condition.

You can also add new parameters by setting the new parameter in the left part equation, the operator must be set to -> and in the right part equation input the value of your new parameter (ex. Xnew -> 0.4). After you input the new parameter you can used it in a new criteria immediately (ex. Vmax<= xnew*Lbridge/360). *You can not overwrite the original parameters.*

Saving tabular results

In Result mode, the data contained in the **Results** dialog may be saved to disk using the button **Save**. This saves the data as a simple ASCII file using the name of the structure project and the .out extension as filename. The file is overwritten without warning.

Printing Tabular Results

In Result mode, the data contained in the **Results** dialog may be printed using the **Print** button.

Viewing Graphical Results

See **Toolbox** section and **Results** tab description

Chapter 5

Components and Tips

General Geometric Shape

The General Geometric shape component computes the geometric properties of predefined parametric beams. These properties are computed with respect to an arbitrary coordinate system. The beam drawing may be exported to BEDAS or any other client applications.
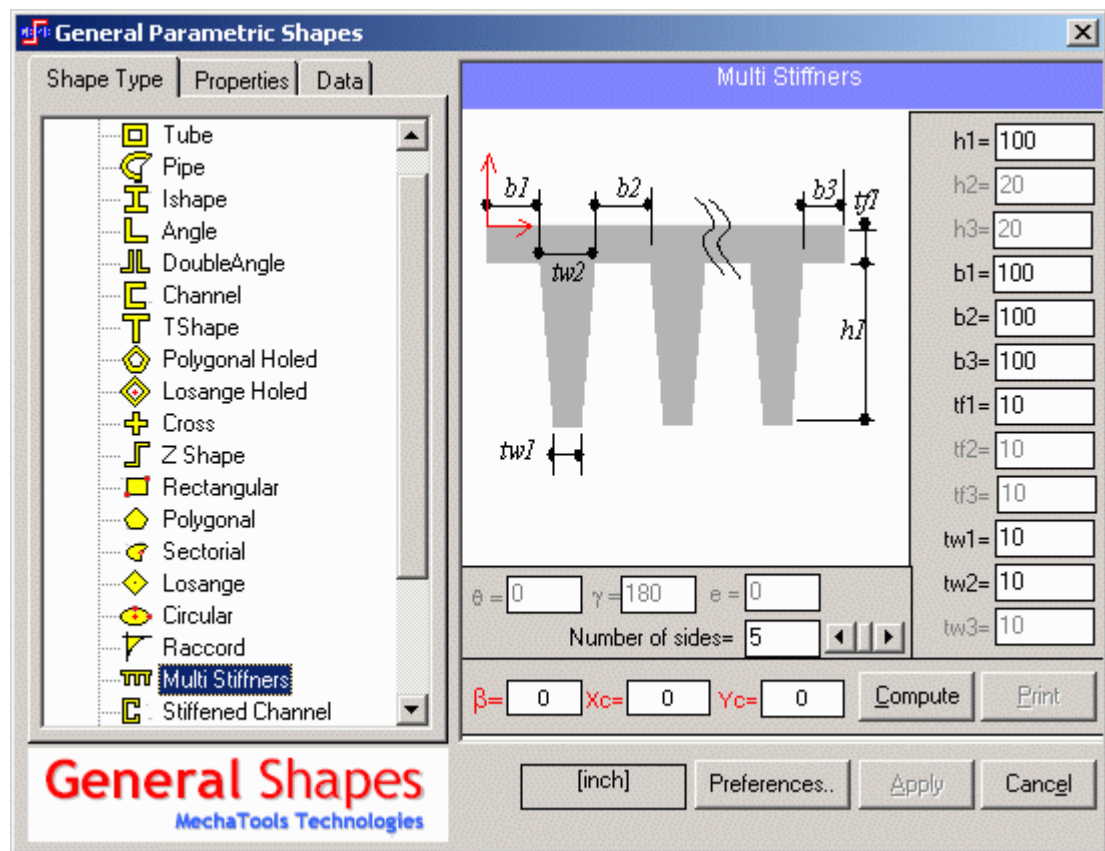
Selecting a beam

To select a beam type, click on one of the predefined beams in the **General Beam** tree of the **Beam Type** tab. Note that the **Free Beam** type is used to create a user-defined beam. When a beam is chosen, a detailed drawing of the chosen parametric beam is displayed. This drawing shows the customizable beam parameters.

Entering parameters

When a beam is chosen, the list of beam specific parameter with default values is displayed on the left and below the beam drawing. Parameters that do not apply to this beam are grayed out and inaccessible. Enter the beam parameters.

Specifying the coordinate system

Parametric beams are defined with respect to a beam specific coordinate system. This coordinate system is shown in red in the detailed drawing of the chosen parametric beam. To compute geometric properties with respect to an arbitrary coordinate system, specify the orientation and position of the coordinate system using the *beta* angle, **Xc** and **Yc** parameters.

Computing Properties

To compute properties, click **Compute**. The geometric properties are displayed in the **Properties** tab. The beam coordinates are displayed in the **Data** tab.

Creating User-Defined Beams

To create a user-defined beam based on an existing parametric beam, repeat the previous steps. After clicking **Compute**, select the **Free Beam** type in the **General Beam** tree. Instead of having a list of beam specific parameters displayed, a tabular editing control is displayed. The cells represent the x y coordinates of the points making up the polyline representing the beam. There are a tools bar buttons to add, insert and remove points or clear all entered data.
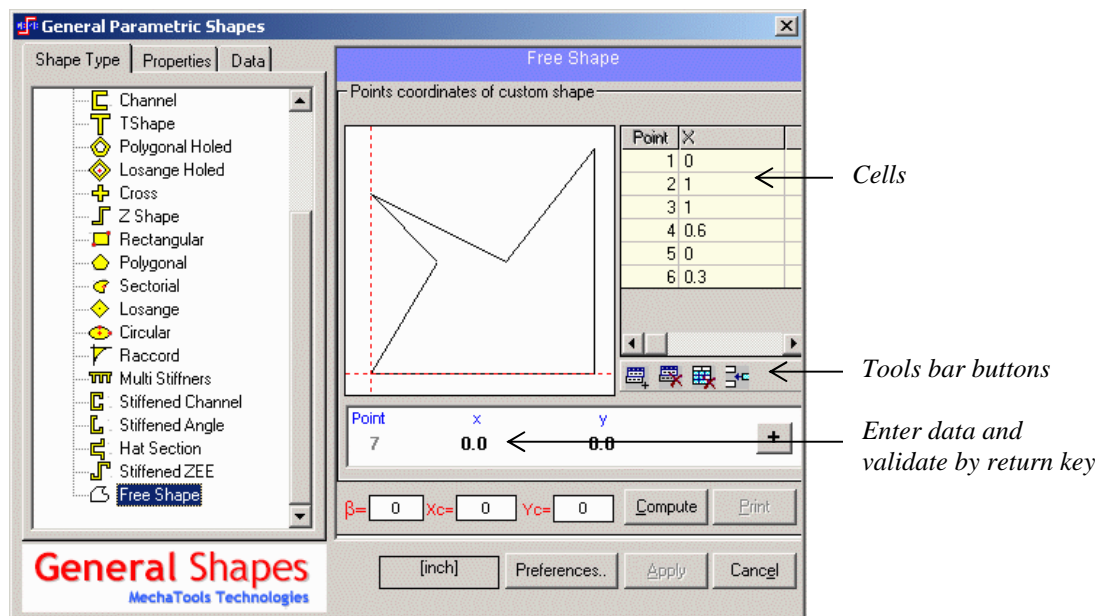
Note that you do not have to base a user-defined beam on an existing parametric beam. You may go directly to the **Free Beam** type and enter data from scratch.

Exporting the Beam

To export the beam geometry to BEDAS (or any other application using this component), click **Apply.**

Printing

If you want print your data and graphic beam, click Print button.

Standards Beams

The General Geometric Beam component displays the geometric properties of predefined standard steel beams. The beam drawings may be exported, using an arbitrary coordinate system, to BEDAS or any other client applications The General Geometric Beam component also supports multi criterion optimal beam selection.

Selecting a beam

To select a beam type, click on one of the standard beams in the **Standard Beam** tree of the **Beam Type** tab. From the beam dropdown, select the specific instance of the beam type. When an instance is selected, a detailed drawing of the chosen beam is displayed. The **Properties** tab contains the properties of the selected beam.

Finding an Optimal Beam

To search for a beam that satisfies a series of criterions, select the **Optimization** tab. From the **Beams** tab, select the types of beams you want to search. From the **Constraints** tab, select the applicable constrains. This is done by checking one or more constraints and specifying the lower and upper value for each constraint. Click **Optimize** to search for the beams. Click on the **Results** tab to view the properties of all beams that satisfy the specified constraints. The valid beams returned and selected using the beams dropdown.
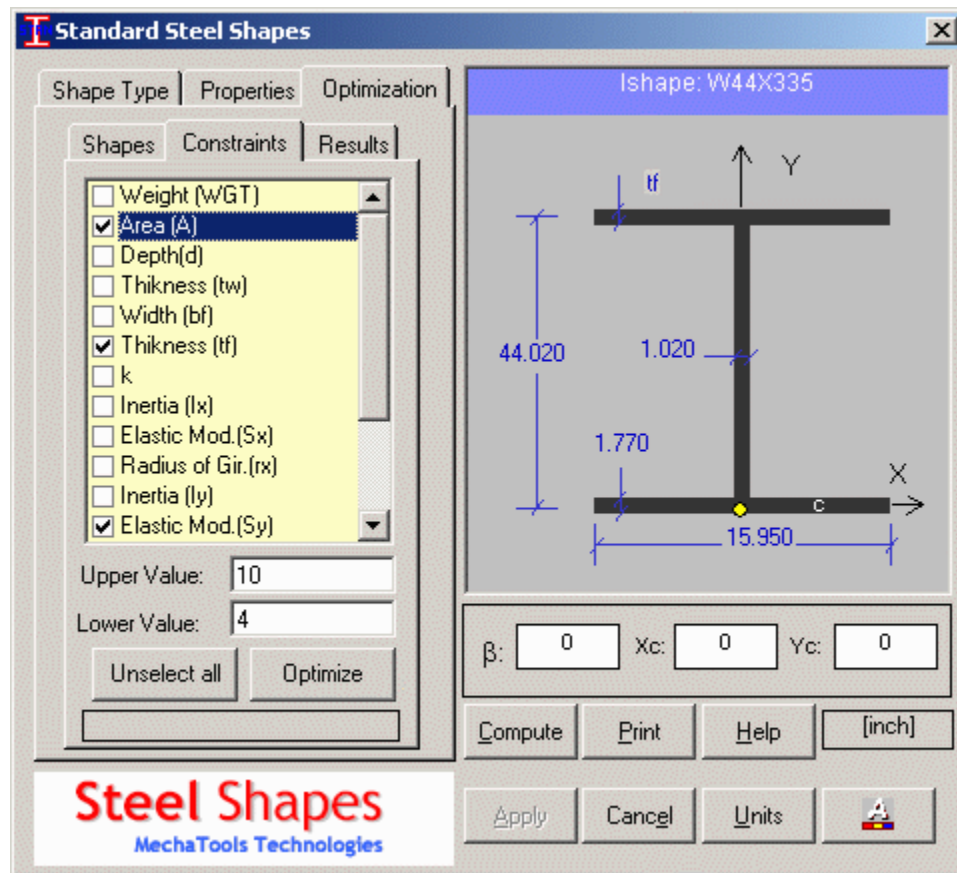
Exporting the Beam

To export the beam to BEDAS (or any other application using this component), specify the coordinate system, transform the beam to the new coordinate system then click **Apply**.

Specifying the coordinate system

Standard beams are defined with respect to a beam specific coordinate system. This coordinate system is shown in black in the detailed drawing of the chosen beam. To move the beam to a new coordinate system before exporting it, specify the orientation and position of the coordinate system using the angle *beta*, **Xc** and **Yc** parameters.

Transforming the beam to the new coordinate system

Click **Compute**.

Printing

If you want print your data, graphic beam and return in report, click Print button (same as the general beam component)

Beam Structure Optimization

Optimization, give you many possibilities, the principal one is, that you can get one beam or more, by giving some constraints. Constraints can be applied to all the parameters and properties of the beam, to apply constrain to a parameter (ex. *Area*), you must input the upper and lower values, so the component looks for:

Lower value <Area<Upper value and so for the other parameters. The other type of constraints is the type of beam, you must select one or more type of beams to applied the previous parameters constraints. If all necessary constrains are applied, press the **Optimize** Button, to see results press the **Results** Tab. See figure in **Finding an Optimal Beam** paragraph.

Results can contain none or many beams that respect the constrains. List of results is given in the list box, in the **Results** Tab. From the list box, if you select a beam, you can see the corresponding properties in the list of properties
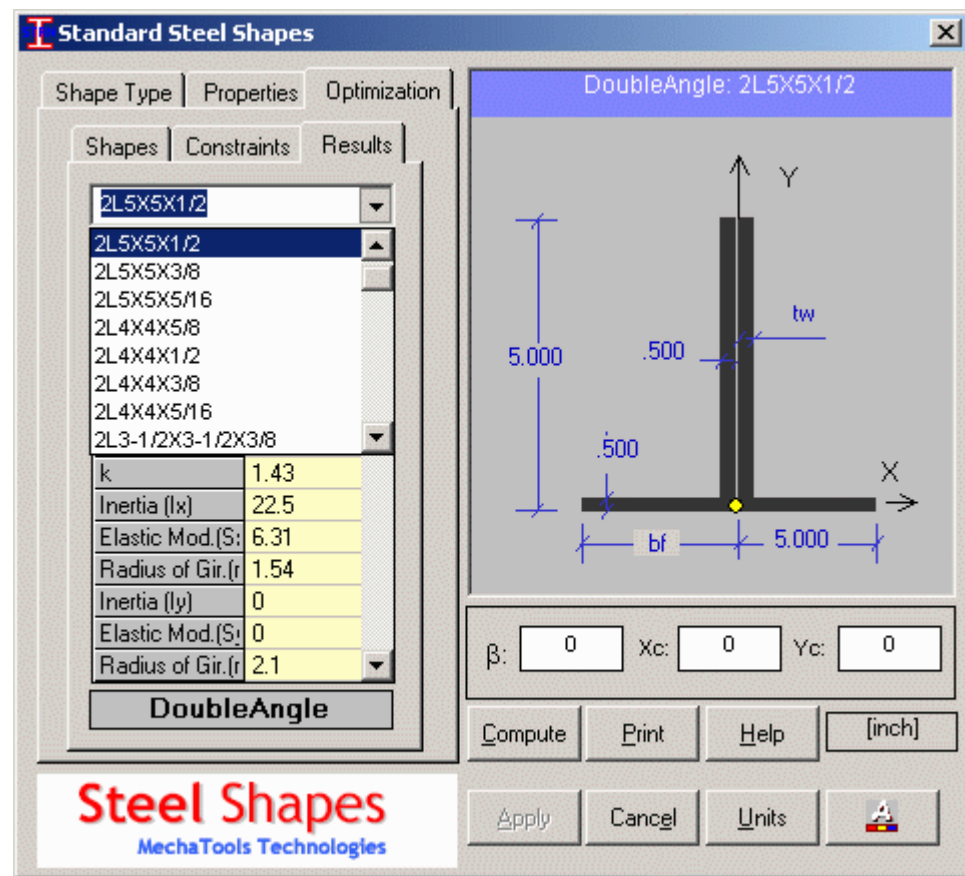
The other utility of the optimization feature is: if you draw or import a custom structure, when you perform properties calculation, you can get the corresponding standard beam

which respect some bounded dimensions and properties of the original structure, this give you the possibility to replace the complicate (expensive) structure with an other standard structure and not expensive.

Utilities

From the toolbox, select **Utilities** tab. The utilities tab contains some utilities like, a useful Units converter component, which allows you the possibility to convert units at any time without searching in your handbook and other documents.

Units converter
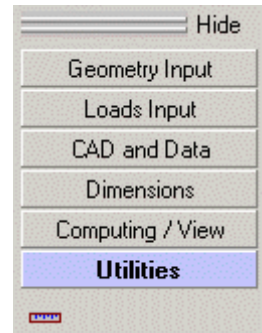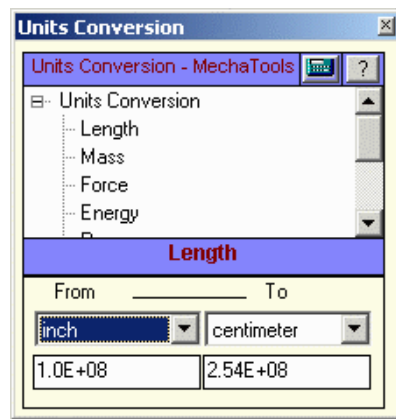
Chapter 6

Techniques and Conventions

Exact Dynamic Finite Element Formulation - Theoretical Background
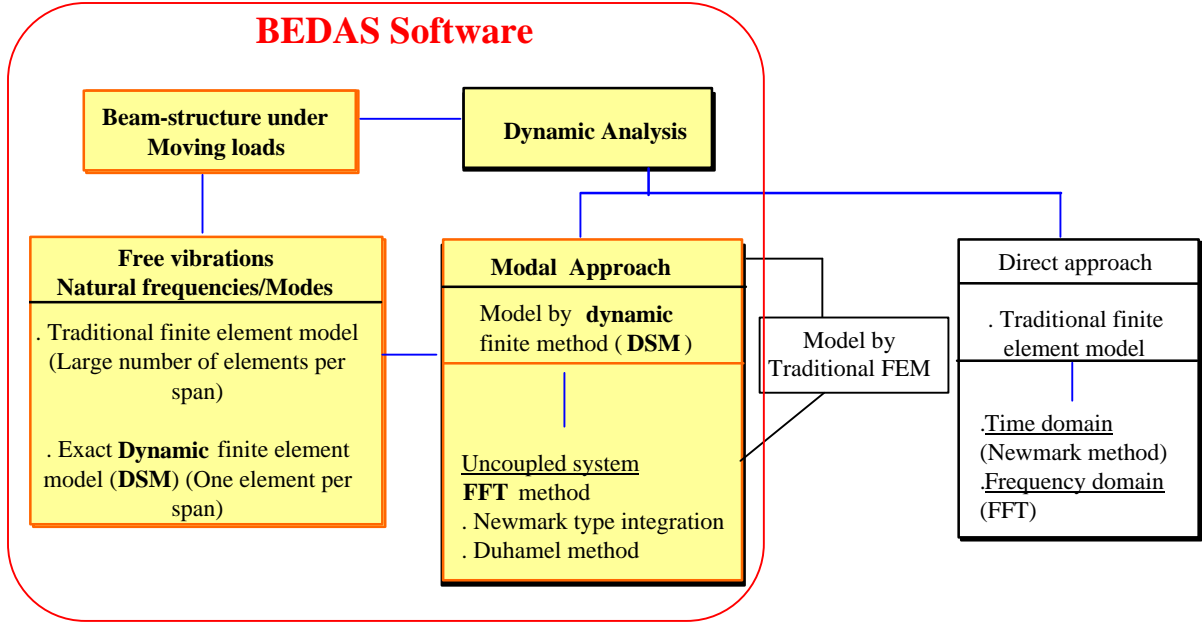
Variational Formulation

The weak formulation of virtual work expression of the transverse vibration of Euler-Bernoulli beam under moving loads for one element, is (we can do the same approach for the axial vibration as the case of Bedas):

$$W^e = \int_0^L \left(dw\, rA\, w_{,tt} + dw\, c\, w_{,t} + dw_{,xx}\, EI\, w_{,xx} \right) dx - \int_0^L dw\, F(x,t)\, dx = 0 \quad \forall dw \qquad (1)$$

In free vibration, we take c and $F(x,t)$ equal to zero.

$W^e = \int_0^L \left(dw_{,xx} EI\, w_{,xx} + dw\, rA\, w_{,tt} \right) dx$ Then, the expression of W^e is reduced to the following form:

$$W^e = EI \langle d\mathbf{a} \rangle \left(\left[\{P_{,xx}\}\langle P_{,x}\rangle \right]_0^l - \left[\{P_{,xxx}\}\langle P \rangle \right]_0^l \right) \{\mathbf{a}\} \qquad (2)$$

We can now express the displacement w and dw in terms of the nodal displacement using:

$$w(x) = \rangle P(x,a)\langle [P_n]^{-1}\{w_n\} = \rangle N(x,a)\langle\{w_n\} \qquad \text{(Idem for } dw) \qquad (3)$$

We can write the discretized form of W^e as:

$$W^e = \langle dw_n \rangle [k(w)] \{w_n\} \tag{4}$$

Where $[k(w)]$ is the dynamic stiffness matrix.

Free Vibration

After assembling elementary dynamic stiffness $[k(w)]$ and elimination of rows and columns corresponding to the boundary conditions, the free vibration problem becomes: $[K_d(w)]\{X_n\}=\{0\}$. The solution of this non-linear eigenvalue problem is performed using Wittrick & Williams algorithm. Projection of the dynamic equation in the modal space is easily obtained and given by:

$$\ddot{y}_j(t) + 2x_j w_j \dot{y}_j(i) + w_j^2 y_j(t) = p_j(t) \tag{5}$$

Where $p_j(t)$ is the generalized load.
For dynamics analysis you can input the damping ratio if it is present in your structure. Damping coefficient is a percentage of the critical damping of the structure materials. Damping is considered a proportional viscous damping:

$$x_i = \frac{a}{2w_i} + \frac{bw_i}{2}$$

The dynamic solution can be obtained by transforming the modal equation into frequency domain, using FFT technique:

$$\left(-\Omega^2 + 2i\Omega x_j w_j + w_j^2\right) Y_f(\Omega) = P_f(\Omega). \tag{6}$$

To determine the undamped frequency w_n for the continuous beam-structure, the number $J(w^*)$ of frequencies of this structure exceeded by a fixed trial frequency w^*, is given by:

$$J(w^*) = J_0(w^*) + s\{K_d(w^*)\} \tag{7}$$

Where $J_0(w^*)$ is the number of frequencies that are exceeded by w^* if all degrees of freedom of the structure are clamped (displacements being constrained to zero) and $s\{K_d(w^*)$ is the number of negative diagonal elements in the triangulated dynamic stiffness $[K_d(w^*)]$ evaluated at $w=w^*$. We use an iterative method like the bisection method, in order to converge to the true frequency. Corresponding to each frequency w, mode shapes of the rth span are defined by:

$$f_{jr}(x_r) = A_{jr} \cos(a_{jr}x_r) + B_{jr} \sin(a_{jr}x_r) + C_{jr} \cosh(a_{jr}x_r) + D_{jr} \sinh(a_{jr}x_r) \tag{8}$$

where f_{jr} is the jth mode shape of the rth span (Figure 1) and x_r is the local coordinate of the span r. We define the jth mode shape of the whole structure by:

$$\Phi_j(X_r) = f_j(x_r), \text{ Where: } X_r = x_r + \sum_{i=1}^{r-1} l_i \tag{9}$$

l_i is the length of the span i
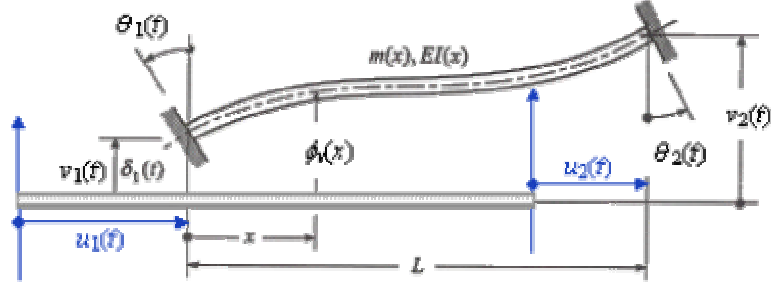
Modal Approach

The representation of equation (1) in modal space leads to a set of uncoupled relations. For each element, the modal representation of transverse deflection is:

$$w(x_r,t) = \sum_{j=1,n} \Phi_j(X_r)y_j(t) \tag{10}$$

Using equations (10) and the orthogonal properties of modes in equation (1), we obtain:

$$\ddot{y}_j(t) + 2\mathbf{x}_j\mathbf{w}_j\dot{y}_j(i) + \mathbf{w}_j^2 y_j(t) = \Delta_j \int_0^L \Phi_j(x)\sum_{i=1,nl}\mathbf{d}(x-x_i)F_i(x,t)dx = p_j(t) \tag{11}$$

Where:

$$\Delta_j = 1/M_j^2 = \cfrac{1}{\displaystyle\sum_{r=1,span}\int_0^{l_r} m_{l_r}\mathbf{f}_{jr}^2(x)dx} \tag{12}$$

Frequency Domain Response

The solution can be more easily obtained by transforming equation (11) into the frequency domain, using Fourier transform technique:

$$P_f(\Omega) = \int_{-\infty}^{+\infty} p(t).e^{-i\Omega t}dt \ ; \ Y_f(\Omega) = \int_{-\infty}^{+\infty} y(t).e^{-i\Omega t}dt \tag{13}$$

$$i\Omega Y_f(\Omega) = \int_{-\infty}^{+\infty} \dot{y}(t).e^{-i\Omega t}dt \ ; -\Omega^2 Y_f(\Omega) = \int_{-\infty}^{+\infty} \ddot{y}(t).e^{-i\Omega t}dt \tag{14}$$

Using equations (13) and (14), equation (11) becomes:

$$\left(-\Omega^2 + 2i\Omega\mathbf{x}_j\mathbf{w}_j + \mathbf{w}_j^2\right)Y_f(\Omega) = P_f(\Omega) \tag{15}$$

Thus the solution in the time domain is

$$y_j(t) = \frac{1}{2p} \int_{-\infty}^{+\infty} Y_f(W) \, e^{iWt} \, dW$$

$$\dot{y}_j(t) = \frac{1}{2p} \int_{-\infty}^{+\infty} i \, W Y_f(W) \, e^{iWt} \, dW$$ (16)

$$\ddot{y}_j(t) = \frac{1}{2p} \int_{-\infty}^{+\infty} - W^2 \, Y_f(W) \, e^{iWt} \, dW$$

We should choose the time period T_2 large enough to cover the response characteristics of the system: $Dt = T_2 / N;\ T_2 = T_1 + T;\ \ \ \ T_1 = 2\pi/w_1$

Where T_1 is the fundamental period of the structure and T is the period of the excitation load. In order to discretise the pulsation interval Ω, we use $\Delta\Omega = 2p\,/\,T_2$

With $0 = \Omega = \Omega_{max} = 2p/Dt$ where Ω_{max} is the Nyquist pulsation. The choice of Dt is such that $f_s = 1/2Dt$ where f_s is the highest frequency present in the signal and participating in the response of the structure. One should verify that: $Dt = 1/2f_{max}$

The discrete representation requires a judicious choice of T_2 and N, which defines Dt, $D\Omega$. One may remark that the time and pulsation are divided into N steps.

To obtain a very good approximation of the displacements, velocities and accelerations, we must evaluate the generalized loads $p_j(t)$ in exact procedure, because the transfer function between loading $p_j(t)$ and displacement $y_j(t)$ is generated analytically by equation (15).

Representation of the Excitation Loading

In the following cases, we assume that the level of the moving or fixed impulse load F_i could be given in the general form as: $F_i = F_i(t)$. We assume the same equation for the distributed moving or fixed load q_i.

Moving Multi-axle Loads

The generalized load for the moving multi-axle load case (Figure 1) is:

$$p_j(t) = \Delta_j \int_0^L \Phi_j(x) \sum_{i=1,nl} d(x - x_i) F_i(x,t) dx = \Delta_j \sum_{i=1,nl} \Phi_j(\bar{x}_i) F_i(\bar{x}_i, t)$$ (17)

Where $x = v_i\, t$, and v_i is the speed of the moving load F_i. If the load F_i is on the span s, we have:

$$\Phi_j(\overline{x}_i) \, F_i = \textbf{\textit{f}}_{js}(x_s) \, F_i \tag{18}$$

$$x_s = \overline{x}_{nc} - \sum_{r=1}^{nc-1} l_r$$

$\overline{x}_{nc}$ is given in function of the position of the first load by

$$\overline{x}_{nc} = \overline{x}_1 - \sum_{j=1}^{nc-1} a_j \tag{19}$$

With a_j is defined as the relative position between the loads F_j and F_{j-1} (Figure 1).

Figure 1: 2D framed bridge under multi-axle moving loads

Moving Distributed Loads

The generalized load is given by:

$$p_j(t) = \Delta_j \int_0^L \Phi_j(x) q_i(x,t) dx = \Delta_j \sum_{p=1,nspan} \left(\int_0^{l_p} \textbf{\textit{f}}_{jp} \sum_{i=1,nl} q_i(x,t) dx \right) \tag{20}$$

If the distributed load q_i is on the span s, we have:

$$\int_0^{l_s} \boldsymbol{f}_{js} \sum_{i=1,nl} q_i(x,t)dx =$$

$$\frac{q_j}{\boldsymbol{a}_{js}} up\left(A_{js} \sin\left(\boldsymbol{a}_{js} x\right) - B_{js} \cos\left(\boldsymbol{a}_{js} x\right) + C_{js} \sinh\left(\boldsymbol{a}_{js} x\right) + D_{js} \cosh\left(\boldsymbol{a}_{js} x\right)\right)\Big|_{x_{lo}}^{x^{up}} \qquad (21)$$

Where x_{up}, and x_{lo} are given in Figure 2.
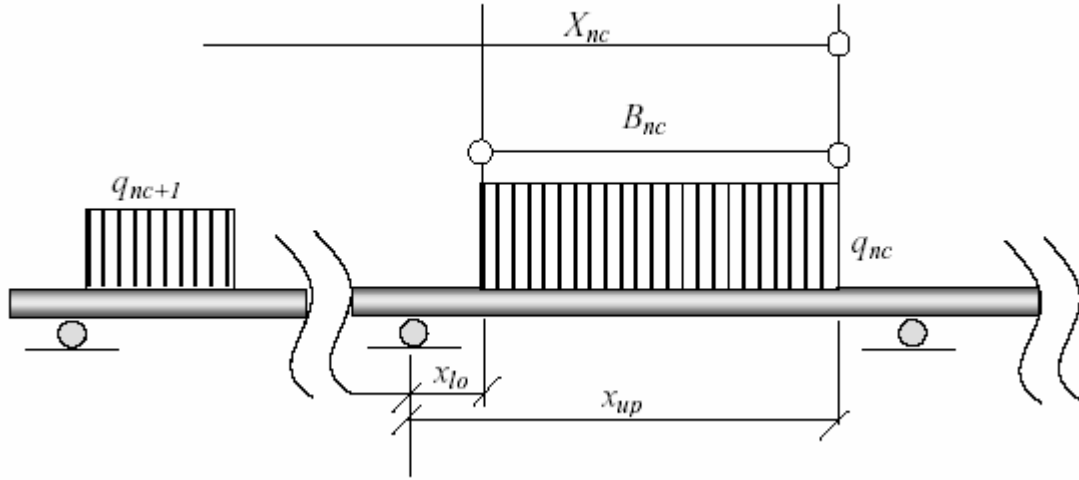
Figure 2: The upper and lower boundary of the integral of equation 21. Case of distributed moving loads

Internal Forces Computation

Because the approximations of the modal responses (frequencies and mode shapes) are very good using a dynamic stiffness approach, the internal forces are obtained in a very good way for any structure in the span and for any time.
Bending moment in the rth beam at x_r location and at any time t is given by:

$$M(x_r,t) = -\frac{\partial^2 w(x_r,t)}{\partial x^2} EI_r = -EI_r \sum_{j=1,n} \boldsymbol{f}_{jr,xx}(x_r)y_j(t) \qquad (22)$$

Shearing force in the rth beam at x_r location and at any time t is given by:

$$T(x_r,t) = -\frac{\partial^3 w(x_r,t)}{\partial x^3} EI_r = -EI_r \sum_{j=1,n} \boldsymbol{f}_{jr,xxx}x(x_r)y_j(t) \qquad (23)$$

It will be necessary to include more modes to define the internal forces to any desired degree of accuracy than to define the displacements; their contributions become more significant for moment

response and even more significant for shear. Therefore, one should be careful when limiting the number of lower modes in estimating response.

The Flexure formula

The fiber stresses at the extreme (i.e. top and bottom) fibers may be written, respectively, as:

$$s_x^{top}(x_r,t) = -\frac{M(x_r,t)}{I_r} y_{max} = -\frac{M(x_r,t)}{S_{1r}}$$

$$s_x^{bot}(x_r,t) = -\frac{M(x_r,t)}{I_r} y_{min} = -\frac{M(x_r,t)}{S_{2r}}$$

Where, I is the moment of inertia about neutral axis and the quantity S is called the elastic section modulus of the r current beam cross-section. All these quantities are computed in BEDAS, and can be used in the design verification dialog, including the maximum dynamic values in displacements, rotations, velocities, accelerations, internal efforts, dynamic amplification factors and the maximum dynamic responses.
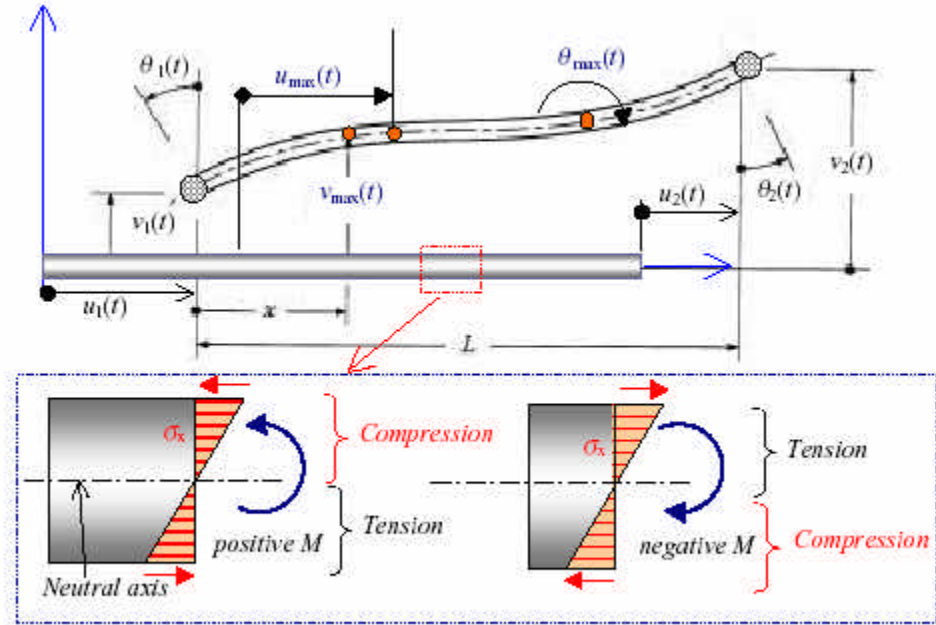
Dynamic amplification factor (DAF) - DLF

In Bedas the dynamic amplification factor is computed as fellow:

DLF (or DAF) = maximum dynamic response/ maximum static response

Static (Quasi-Static) Response

With the exact dynamic stiffness approach, we can also obtain easily the static response in very good agreement. We assumed that the inertial and damping effects vanish, so the modal equation (11) becomes:

$$\boldsymbol{w}_j^2 y_j(t) = \Delta_j \int_0^L \Phi_j(x) \sum_{i=1,nl} \boldsymbol{d}(x - x_i) F_i(x,t) dx = p_j(t)$$

Moreover, the modal static response is: $\boxed{y_j(t) = \dfrac{p_j(t)}{\boldsymbol{w}_j^2}}$

With pj is the generalized force of the static load $F(x,t)$. The physical response $w(x)$ is given by equation (10). We take the example of a simple pinned-pinned beam under a constant force F applied at the mid-beam at $x=L/2$, the mode shape and the frequency of mode j are given by:

$$\boldsymbol{f}_j(x) = \sin\left(\frac{j\boldsymbol{p}x}{L}\right) \text{ And } \boldsymbol{w}_j = \left(\frac{j^2}{L^2}\boldsymbol{p}\right)\sqrt{\frac{EI}{m_L}} \quad j=1, 3, 5, 7, \dots,$$

And

$$p_j(t) = \int_0^L \boldsymbol{f}_j(x)\boldsymbol{d}\left(x - \frac{L}{2}\right)F dx = \boldsymbol{f}_j(\frac{L}{2}).F = F\sin\left(\frac{j\boldsymbol{p}}{2}\right)$$

From equation (12) $M_j^2 = L/2$. We obtain:

$$y_j(t) = \left(\frac{1}{m_l \, L/2}\right)\left(F\sin\left(\frac{j\boldsymbol{p}}{2}\right)\right)\left(\frac{L^4 m_l}{j^4 \boldsymbol{p}^4 EI}\right) = \frac{2FL^3}{EI\boldsymbol{p}^4}\frac{1}{j^4}\sin\left(\frac{j\boldsymbol{p}}{2}\right)$$

Finally, from equation (18) the physical response $w(x)$ is given by:

$$w(x) = \sum_{j=1}^{n\,\mathrm{mod}\,e} \boldsymbol{f}_j(x) \cdot y_j(t) = \sum_{j=1}^{n\,\mathrm{mod}\,e} \frac{2FL^3}{EI\boldsymbol{p}^4}\frac{1}{j^4}\sin\left(\frac{j\boldsymbol{p}}{2}\right)$$

And the maximum static response of the beam is obtained at $x=L/2$:

$$w(\frac{L}{2}) = \frac{FL^3}{48.7EI}\left(1 + \frac{1}{3^4} + \frac{1}{5^4} + \frac{1}{7^4} + \dots\right)$$

If we take the first seven participate mode ($j=7$), we find

$$\boxed{w(\frac{L}{2}) = \frac{FL^3}{47.99EI}}$$

So, if j (number of mode) is high, $w(L/2)$ converge to the analytical value:

$$w(\frac{L}{2}) = \frac{FL^3}{48EI}$$

In dynamic problems under the influence of base excitations, usually the number of modes considered must contribute to a total mass participation factor of at least 80-85% of the system mass in the direction of the base motion.

For harmonic and random vibration problems, in addition to the 80-85% mass participation factor requirement, the range of natural frequencies considered for the analysis must cover the highest frequency in the excitation.

Report Problems or Errors

To report problems or errors, please record the exact steps that lead up to the problem so that we may reproduce the condition.

Email or fax these steps to our technical support. Any additional information that may be relevant to locating or verifying the problem is appreciated.

Examples:

1.Mode shapes and frequencies calculation of 2D beam structure
(In folder ../examples/ Struct2D.BDS)

$EI=5.0 \times 10^6$ N.m^2
$EA=9.0 \times 10^8$ N
m=35 Kg/m

Dynamic MOMENT at time: 3.2(s)
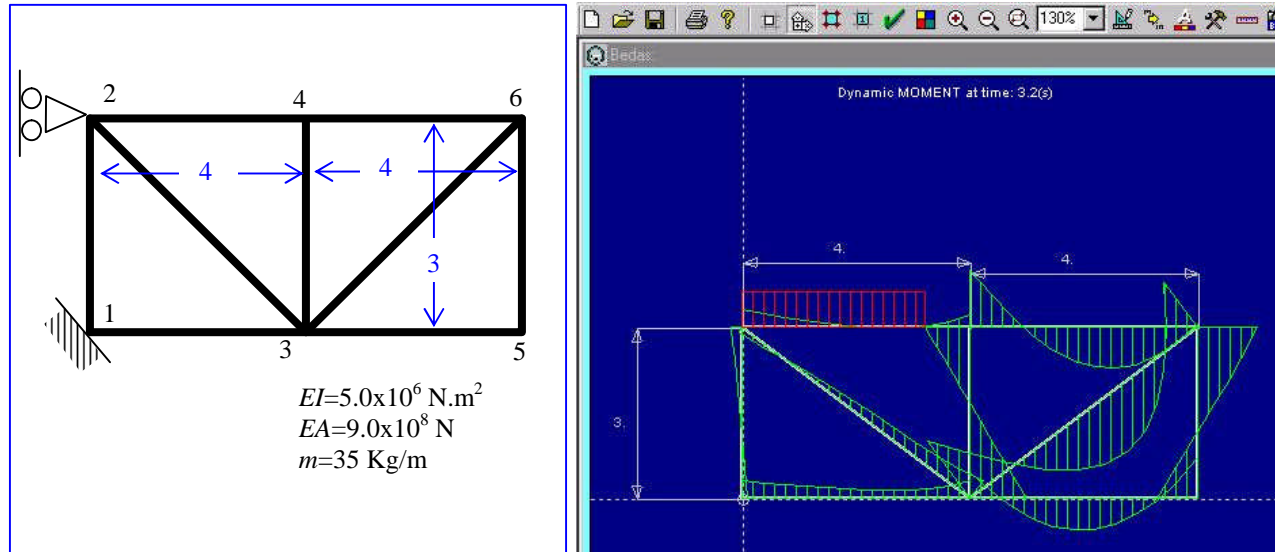
		Mode number				
Standard Finite element		I	2	3	4	5
NNODES	NDOFS	Frequency (Hz)				
6	14	26,373	52,291	62,121	72,541	86,218
9	45	26,796	38,711	46,050	52,116	63,259
24	72	26,088	39,592	48,577	52,583	66,519
33	99	25,823	39,592	48,899	52,569	66,761
43	126	25,898	39,556	48,992	52,537	66,779
51	153	25,630	39,528	49,030	52,513	66,771
60	180	25,587	39,509	49,050	52,497	66,761
69	207	25,471	39,447	49,092	52,442	66,715

		Mode number				
BEDAS		I	2	3	4	5
NNODES	NDOFS	Frequency (Hz)				
6	14	25,4676	39,4660	49,1172	52,472	66,7489

NNODES=Number of nodes, NDOFS= *Number of degrees of freedom*

2. Multi-span beam under a moving load (in folder ../examples/ 3spans.BDS)

Example of continuous beams under a single moving load. We study the dynamic responses for speed v=35.57 m/s, using one element per span (only three elements for the bridge).
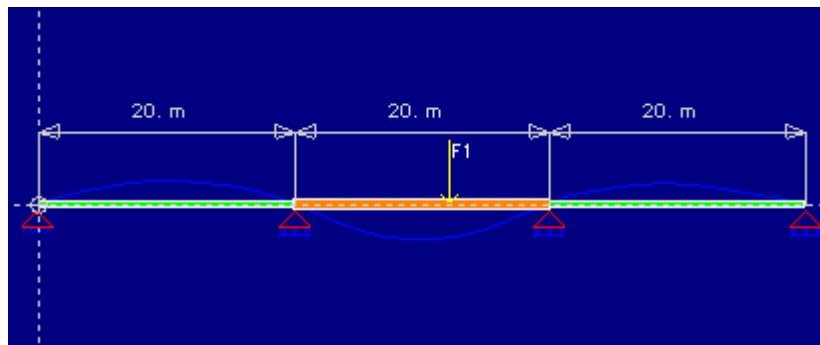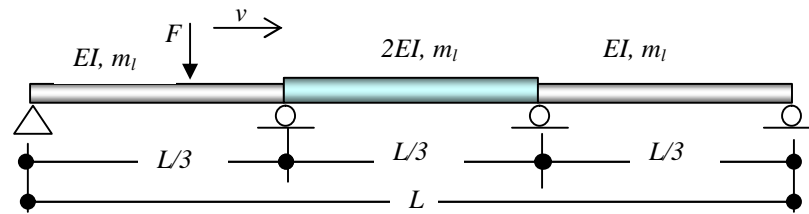
Structure properties	Mode number	Frequencies (Hz)
	1	6.204
$L = 60\ m$	2	7.581
	3	11.974
$m_l = 1.0\ \mathrm{x}10^3\ kg/m$	4	21.163
$A = .51\ \mathrm{x}10^{-2}\ m^2$	5	24.207
	6	26.439
$EI = 1.96\ \mathrm{x}\ 10^9\ N.m^3$	7	37.282
$E = 10.48\mathrm{x}10^{\,10}\ N/m^2$	8	53.579
	9	56.642
$F = 9.8\ \mathrm{x}10^3\ N$	10	64.130

Dynamic Analysis response in Beam 2 at local x=10.00 [m] : V(10.,t) / 26.824E-05 [m]

Dynamic Analysis response in Beam 3 at local x=10.00 [m] : V(10.,t) / 48.864E-05 [m]

3.Dynamic of Multi-Span Bridges under the effects of high speeds moving train

We study the dynamic response of three different bridges under the effects of the high speeds moving train (TGV).
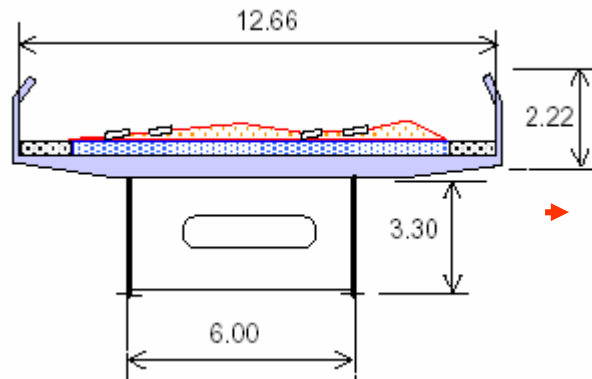
Geometry of typical bridge structure
(dimensions in [m] for La Lys Bridge).
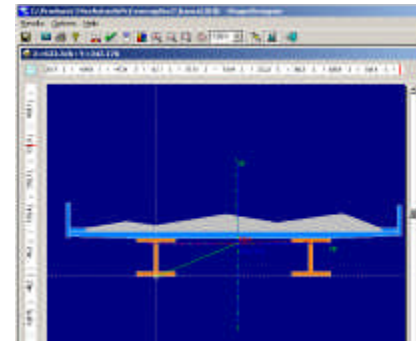
For the geometric properties of the composite section, we used our ShapeDesigner software
www.mechatools.com

The vehicles used in these analyses are the French TGV train, which composed of two cars (the power car and trailers for each car) as shown in figure. In this case, the series of loads has 60 axles of 17 tones for each axle, and the train speed varies from 150 km/h to 400km/h in order to determine the critical and sub-critical speeds and the vehicle-bridge response characteristics. The elastic modulus used in the three cases of bridges is 2.1×10^{11} MPa.
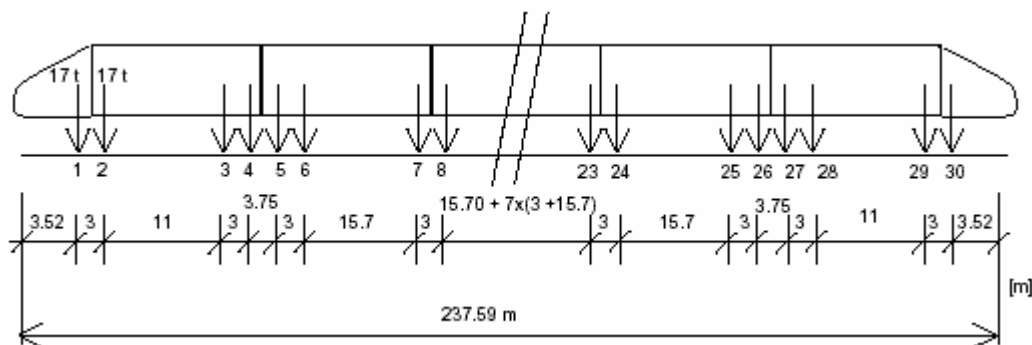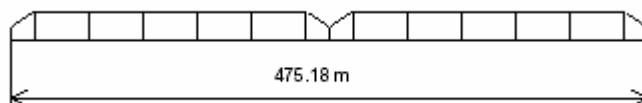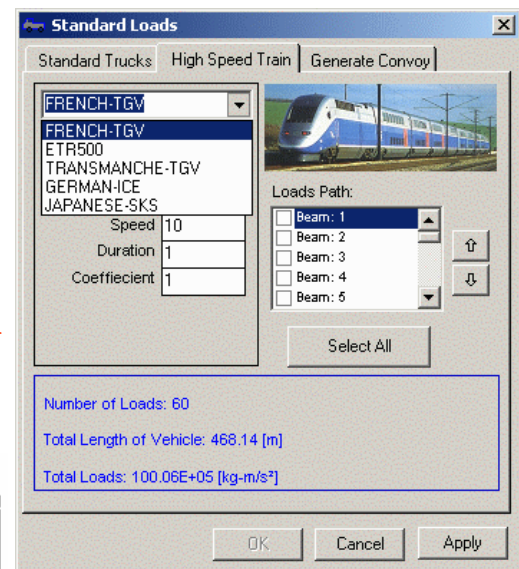
The dimensions of the one car French TGV

3.1. LA LYS Bridge (in folder ../examples/ LYS.BDS)

Problem Description

The LA LYS Bridge is a fast speed train (TGV) railway bridge. The viaduct is a continuous bridge of three spans (30.20 m-53.20m-30.20m), of 113.6 m length and four structures kinds. Structure 3 is a cracked concrete region, and the modular ratio between steel and concrete elastic modulus taken in this example is 10 under live loads. The coefficient of damping is taken equal to 0.5% of the critical damping; the train speed varies from 150 to 400 km/h and the viaduct is simply supported.
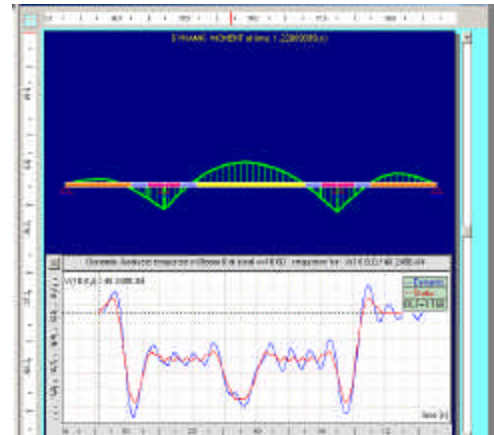
Sections	1	2	3	3 cracked	5
ρ [kg/m^3]	51610.0	45549.0	42765.0	80917.0	48794.0
A [m^2]	0.67352	0.7774	0.83948	0.41784	0.72018
J [m^4]	0.6500	0.6605	0.6622	0.6622	0.6533
Iz[m^4]	0.9144	1.3067	1.5249	0.8587	1.2572

Properties and material data of the LA LYS bridge structure
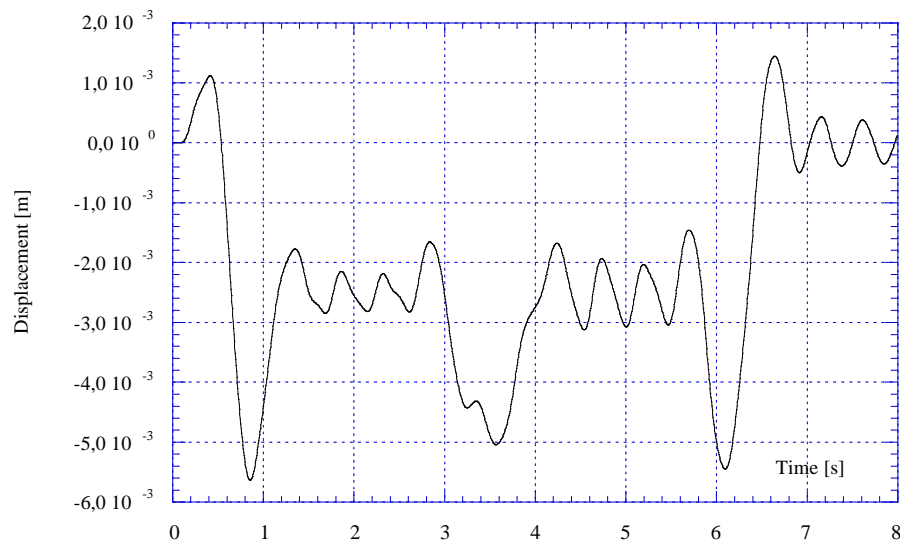
Mode number	Frequencies (Hz)
1	2.1417
2	4.7135
3	5.3843
4	7.9452
5	14.774
6	18.463
7	26.964
8	37.076
9	40.660
10	44.419

Dynamic response in central mid-span, v=300 km/h. Vertical displacement, LA LYS Bridge.

3.2. BANCEL Viaduct (in folder ../examples/ Bancel.BDS)

Problem Description

The BANCEL Viaduct is a fast speed train (TGV) railway bridge. The viaduct is a continuous bridge of five spans, a length of 218 m and 3 structures kinds. Structure 3 is a cracked concrete region, and the modular ratio between steel and concrete elastic modulus taken in this example is 10 under live loads. The coefficient of damping is taken equal to 0.5% of the critical damping, the train speed varies from 150 to 400 km/h and the viaduct is simply supported.
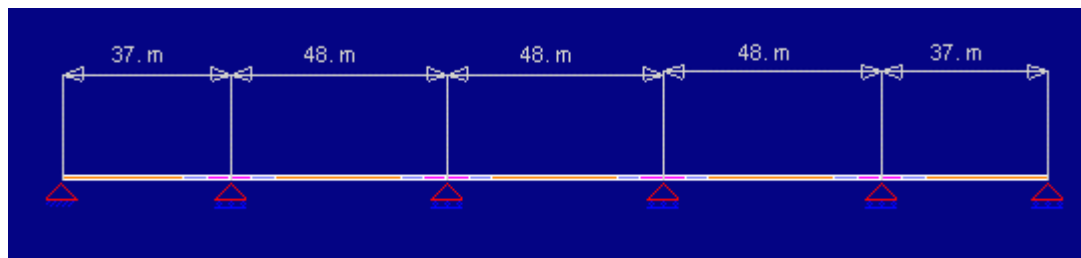
Sections	1	2	3
ρ [kg/m^3]	6447.5	80495.9	60622.4
A [m^2]	4.532	0.363	0.482
J [m^4]	0.410	0.410	0.410
Iz[m^4]	1.215	0.665	0.964

Properties and material data of the Bancel bridge structure

Nodes

Sections

Mode number	Frequencies (Hz)
1	2.108
2	2.634
3	3.382
4	4.341
5	4.588
6	7.683
7	9.026
8	10.572
9	13.730
10	14.133
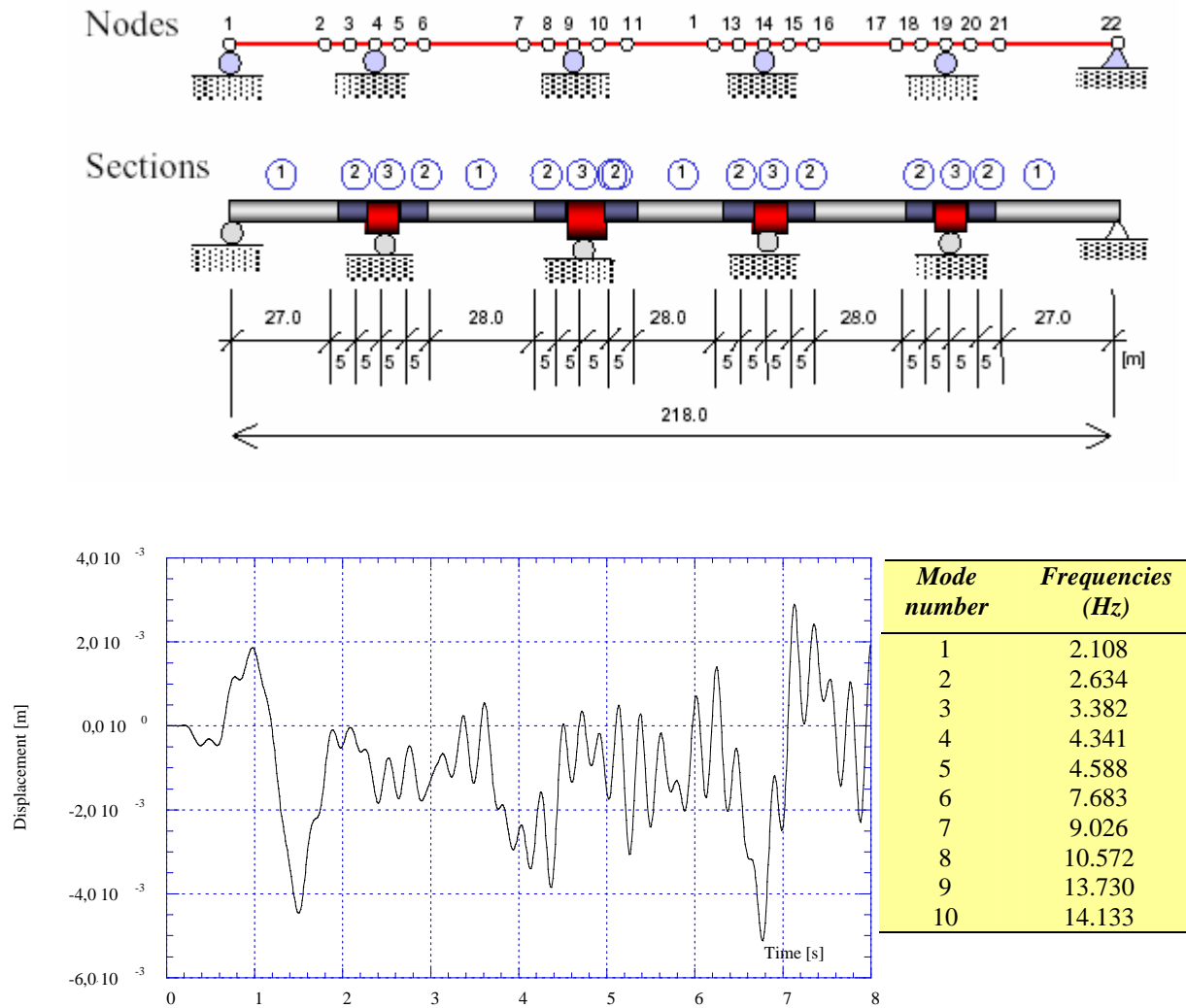
Dynamic response in central mid-span, v=300 km/h. Vertical displacement, BANCEL Bridge.
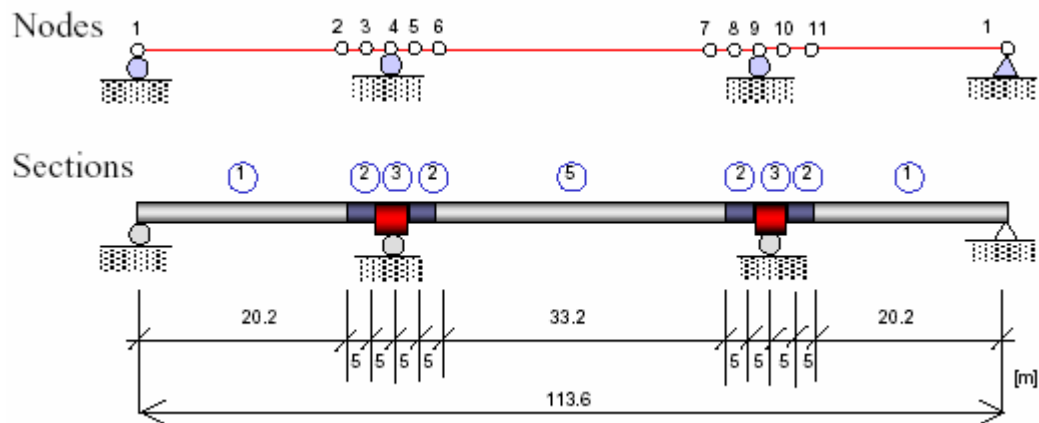
3.3. LA SCARPE Bridge (in folder ../examples/ Scarpe.BDS)

Problem description

The LA SCARPE Bridge is a fast speed train (TGV) railway bridge. The viaduct is a continuous bridge of nine spans, and length of 333 m, 4 structures kinds. In figure below, structures 3 and 4 are cracked concrete regions, and the modular ratio between steel and concrete elastic modulus taken in this example is 10 under live loads. The coefficient of damping is taken equal to 0.5% of the critical damping, the train speed varies from 150 to 400 km/h and the viaduct is simply supported..

Sections	1	2	3	4
ρ [kg/m^3]	46350.0	44930.0	93500.0	88500.0
A [m^2]	0.63	0.65	0.313	0.330
J [m^4]	0.40	0.40	0.40	0.40
Iz[m^4]	0.586	0.686	0.390	0.450

Properties and material data of the LA SCARPE bridge structure

Nodes

Sections

333.0

Mode number	Frequencies (Hz)
1	2.174
2	2.366
3	2.660
4	3.027
5	3.438
6	3.860
7	4.222
8	5.219
9	6.439
10	8.090

Dynamic response in central mid-span, v=300 km/h. vertical displacement, LA SCARPE Bridge.
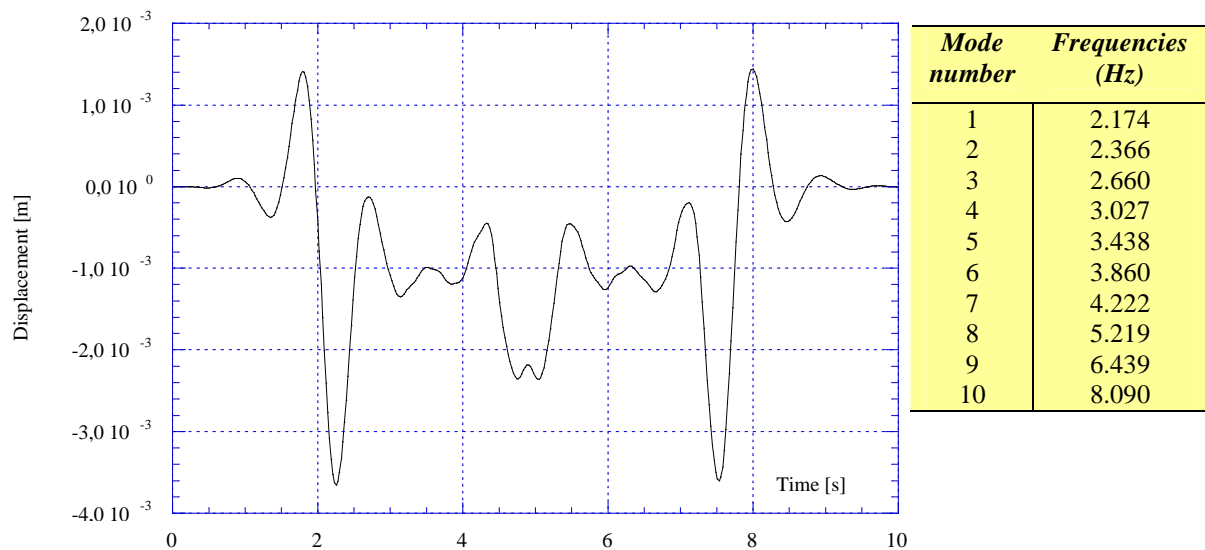
- The critical speeds Vcr are determined by supposing the predominant frequency of loads is function of distance d between the 2 successive axles

  - d=18.7m for one car TVG and $Vcr = d/Ti$, where Ti is a given period

- the critical speed produce the maximum response

	Vcr(Km/h)	fcr(Hz)	Near Mode
Bancel	295 (81.94 m/s)	4.38	4th
Scarpe	290 (80.55 m/s)	5.57	7th
LA LYS	375 (104.16 m/s)	4.30	3rd

Input and Output Files

Input File

The BEDAS input file, is a common ASCII file with extension "**BDS**". Its structure is simple and intuitive. An example is given below: Words in bold are a BEDAS keywords you cannot change any keyword.

```
          M E C H A T O O L S     S T U D I O 1.0

          ****************************************
          *                                      *
          *            BEDAS 4.0                 *
          *                                      *
          *     Dynamic Analyses of Structures   *
          *      And Bridges under Fixed and     *
          *            Moving Loads              *
          *     MECHATOOLS TECHNOLOGIES© 2004    *
          ****************************************
```

Version
BEDAS 4.0
COMPANY
Your Company
CLIENT
Your Client
PROJECT
Your Project
TASK
Your Task
PREJECT-REFERENCE
You References
ENGINEER
ING1
ZOOM
 1.5
XMAX
 12.3577
XMIN
-.975609
YMAX
 11.5447
YMIN
-1.78861
FREQUENCYUP
 0
DAYNAMIC PROPBLEM DURATION
 2.5
DAYNAMIC TIME STEP
.00965251
DAMPING MODE I
 1
DAMPING MODE J
 2
DAMPING KSI I
 0
DAMPING KSI J
 0
LOADS
 1
    LOAD
     1
    LOAD TYPE
          CONCENTRATE
    LOAD LENGHT
     0
    FORCE EXPRESSION *//F(t)*
    -9480

```
            SPEED                    // v
            35.57
            STARTPOSITION          //x0
             0
            DOF
            V
            DURATION FORCE        //duration
             2
            INITIAL TIME FORCE  //F(t)
             0
            TOTAL BEAMS LOADED //Path beam list
             3
             1
             2
             3
        COORDINATES
         2
        00.00000E+00   // x1
        -35.52714E-16 // y1
        110000          // boundary conditions u=0,v=0
        60.00000E+00   // X2
        -35.52714E-16 // y2
        010000          // boundary conditions u free, v=0
        BEAMS
         1
            BEAM
            1
            COLOR
             65280
            TYPE ID
             1
            FILENAME

            EXT_FILENAME

            MASS AREA
             3921560
            MASS DENSITY
             196078
            MASS LINEAR
             999.9978
            IX
             .0187
            AREA
            0.0051
            MATERIAL
            User-Defined
            ELASTIC MODULE
             104800000000
            SHEAR MODULE
             0
            POISSON COEFFICIENT
             .27
            COLMASS
             0
            COLYOUNG
             0
            FIRST POINT
             1
            SECOND POINT
             2
        COTATIONS
         1
            COTATION
            1
            P1
             0
            P2
             2
            DELTAX
             21.1151724466463
            DELTAY
```

```
          5. 7196027057927
              ORIENTATION
              1
LENGTH UNIT
   4
m
MASS UNIT
   1
kg
STRESS UNIT
   2
Pa
FORCE UNIT
   1
N


END
```

Output File

The BEDAS tabulated results Output file, is a commonly ASCII file with extension "**.rst**". Its structure is simple. An example is given below. You can get also graphical results by choosing print from toolbar in the current activate window. And the problem execution information output file is a commonly ASCII file with extension "**.out**"

References

The following references were consulted in the development of BEDAS. You may find them useful for theoretical background.

K. Henchi., M. Fafard, G. Dhatt, M. Talbot, "*Dynamic behaviour of multi-span beams under moving loads*", Journal of sound and vibration, **199**(1), pp. 33-50, 1996.

F. W. Williams, W. H. Wilttrick, "*A general algorithm for computing natural frequencies of elastic structures*", Quarterly Journal of Mechanics and Applied Mathematics, 24, pp. 263-284, 1971.

J. W. Cooley, J.W. Tukey, "*An algorithm for the machine calculation of complex fourier series*", Mathematical Computation **19** , pp. 297-301, 1965.

J. L. Humar, "*Dynamics of Structures*". Prentice-Hall, Englewood Cliffs, New Jersey, 1990.

R. W. Clough and J. Penzien 1993 *Dynamics of Structures*. New York: McGraw-Hill.

K. J. Bathe 1976 *Numerical Methods in Finite Element Analysis.* Prentice-Hall, Englewood Cliffs, New Jersey.

J. M. Biggs 1964 *Introduction to Structural Dynamics*. New York: McGraw Hill

K. Henchi, "*Dynamic Analyses of Structures and Bridges under Moving Loads – Finite elements approach* ", Doctoral theses, Mechanical department, Numerical Models in Mechanics Division. Compiegne University of Technology, France, 1995.

K. Henchi, G. Dhatt, M. Talbot, M. Fafard, "*Comportement dynamique des poutres continues sous les chargements mobiles. Approches semi-analytiques et éléments finis*", Revue européenne des éléments finis, **6**(2), pp. 165-195, 1996.

V. Koulousek, "*Dynamics in Engineering Structures*", Academica Prague, 1973.

L. Fryba 1972 *Vibration of Solids and Structures under Moving Loads*. Groningen: Noordhoff International Publishing.

R. D. Blevins 1979 *Formulas for Natural Frequencies and Mode Shapes*. Van Nostrand.