Program
AMSES Frame2D
User's Manual

Ainet Projektiranje
Aleš Krajnc, s.p.[1]

Ul. heroja Rojška 70
SI - 3000 Celje
Slovenia

Document version: 2.00 β – unrevised version

Updated on 24th April 2002

---

[1] e-mail: info@amses.com

Contents

Preface

The **AMSES Frame2D** 2D program is coming into a mature state. It's first version was issued in 1999 and I am very pleased that it found a large audience all over the world. For many of them it became a *must have* – an indispensable tool.

The new version was enriched with a set of tools that cover the design of steel and reinforced concrete elements. The design was programmed according to the Eurocode 2 (reinforced concrete) and Eurocode 3 (steel) design codes. Thus, the new version is not just a facelift but has many new practical facilities. The analysis results are used in design procedure – the user only needs to provide some additional information to verify the element design.

The story of the **AMSES Frame2D** 2D began several years ago, in the beginning of 1994. Back than I came up with an idea of writing a group of simple and easy to use computer programs that shall make the life of structural engineers easier. Next year, I actually wrote a few simple programs concerned with the reinforced concrete and steel elements design. These programs were warmly accepted by users, however, I felt that this is not what I wanted in the first place. It turned out that programs are good on their own, but they are not so good when they have to communicate with other programs. At that moment I realized that most of the structural engineering software is outdated from the user interface point of view. This led me into a conclusion that integrated computer aided structural engineering design will be possible only if most of the software is written from scratch.

Everyone knows that it is practically impossible to write all structural engineering software from scratch. It wouldn't have much sense, too. Many of the packages available today are very useful, though clumsy. One of the Murphy's laws says something like: "20% of tasks are used 80% of time." According to my experience (I am a structural engineer) and experience of many of my colleagues the Murphy was right. This means that we do not need to rewrite all the software to significantly speed up our work, the 20% will be just fine.

In the beginning of 1997, I gathered a group of developers and supporters and we started our work. The project goal was to write that 20% of the software we use almost all the time. We named our project as AMSES project.

Today, I am proud that our first product has reached the mature state and I wish that you will find this software useful enough to register it and support our further work.

Aleš Krajnc

CHAPTER 1

Getting started

The easiest and the quickest way to get familiar with AMSES Frame2D is to read this chapter. Two examples are revealed in detail. The first example is a simple continuous reinforced concrete beam. A static analysis is performed first and this is followed by the design of one of its elements. In the second example, a more complex case of steel frame that is lied on the reinforced concrete foundation beam is considered. Here, several actions and load cases are applied and designs of several elements are carefully considered.

If you are an experienced computer guru you can skip to section 2 starting on page 11. Others may find next section interesting.

1.1 Folder tree description

During the installation process you have selected the basic folder. The manuals assume you have selected the default option - the Program Files/AMSES folder. Several additional subfolders are also created and their content is briefly presented below.

Frame2D holds all executable and DLL files needed to run the AMSES Frame2D program.

Frame2D/Misc holds several support files, mostly in textual form. These files are startup tips, error messages and some internally used working files.

Frame2D/Help includes all files of the AMSES Frame2D help system.

Frame2D/Examples includes several examples – mostly from the manual.

Common The folder only hosts two subfolders.

Common/DB holds database files, which describe material and cross-sectional characteristics.

Common/Doc stores documents, which are downloaded from the Internet site www.ain sp.si/msgsys.

My Projects is the default location where your files are stored. You can use any other location, of course.
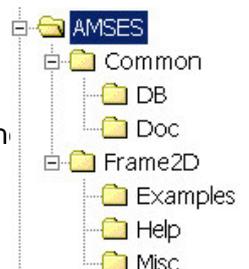
1.2   Setting-up the character set

AMSES Frame2D uses ANSI (Western) character set by default. If you are using some special character set, like Central European, Baltic, Cyrillic, etc. then it is good idea to set it right away:

1. Run the program.

2. Select the Tools|Character Set command and a list of available character sets appears.

3. Select the character set you want to use.

Now the selected character set is active and special characters should appear correctly. The selection is stored permanently, which means you do not need to define the character set ever again.

1.3   File extensions

When a structure you are working on is stored on a disk, two files are created, which differ by their extension. The AMP and AMS extensions are used:

- The AMP extension is bound with a project file. The project file is a relatively short file which stores all file names used in current project.

- The AMS extension is bound with working files - structure files. They hold actual data of the structure: geometry, load, analysis and design results.

When an existing project is opened, the program loads the AMP (project) file first. The AMP file holds information about all working files (structural files) included in the structure. In the next step, all working files are loaded one by one.

1.3.1   Renaming a file

You should keep in mind that working files (AMS extensions) should not be renamed outside the program, e.g. in Explorer. If you change the AMS file name, information that is written in related AMP file will not match the new file name and the program will raise an error when you will try to open it.

In order to rename a file, open the file with AMSES Frame2D and apply the File Save As command.

© 1999-2002 Ainet

Continuous RC Beam – a Quick Tour

The quick tour is a tutorial, which briefly demonstrates some major AMSES Frame2D capabilities. In the tour you will solve your first problem and when the tour completes, you will gather enough knowledge to solve some simple tasks on your own.

2.1 Problem definition

We will analyse and design a simple symmetric reinforced concrete continues beam, which has two fields. The beam geometry is shown on the figure 2.1.
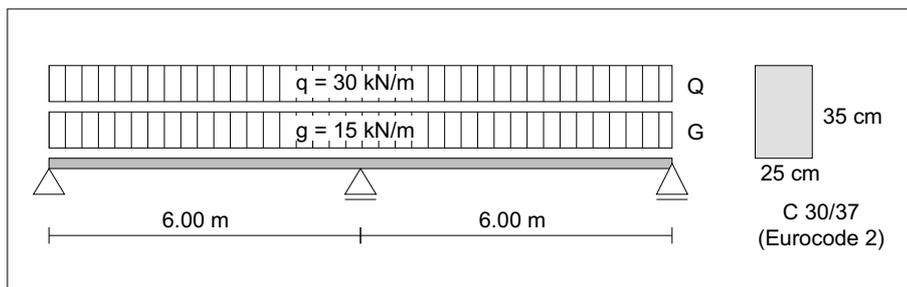
Figure 2.1: Geometry and load of continues beam.

The beam is made of two 6 m wide fields. The leftmost support is fixed, while the others support the beam in z direction only. All supports have the same width of 40 cm. The beam cross-section is rectangle. Its depth is 50 cm and breadth is 30 cm. The beam is made of concrete C30/37, a quality specified by Eurocode 2 [1].

The beam is subject of two uniform loads: permanent load 15 kN/m and traffic load 30 kN/m. The permanent load is member of action (G) and the traffic load is member of action (Q).

These two actions are combined into two load cases. The first load case represents the ultimate limit state and the second represents the serviceability limit state. We named the first one G + Q (uls) and the second G + Q (sls). Results of both load cases will be used later by design process. See table 2.1.

The beam is made of reinforced concrete. When a concrete is subject of long term load it suffers from creep and shrinkage effects. The shrinkage and creep mainly depends on the ambient humidity, the dimensions of the element, the composition of

11

Load case	(G)	(Q)
G + Q (uls)	1.35	1.50
G + Q (sls)	1.00	1.00

Table 2.1: Actions, load cases and load factors

the concrete, the maturity of the concrete when the load first applied, the duration and magnitude of loading. EC2 takes this into account using a simplified approach. It simply reduces the concrete elastic modulus by factor $1 + \phi_c$. Here ϕ_c denotes the creep coefficient, which is obtained from expressions or diagrams in the relevant codes (EC2).

In our case we assume creep coefficient $\phi_c = 2.2$. Since the ultimate limits states are simulations for extreme loading situations, which occur seldom and last only for a short period, they are not prone to the creep effects. On the other hand, the serviceability limit states simulate situations which occur often and can last for longer period. Hence, creep effects must be considered, too. Please, see Eurocode 1 [3] and Eurocode 2 [1] for more details.

2.2 **AMSES Frame2D** solution

The continues beam geometry, materials and its load are well defined and we can start solving the problem. The problem will be solved in several steps:

1. Starting a new document (structure).

2. Defining the geometry.

3. Actions and load cases.

4. Action load.

5. Analysis.

6. Design of individual elements according to EC2.

7. Printout.

2.2.1 Starting a new document

Run the AMSES Frame2D program if it is not already running. Before a new structure can be analyzed, a new document had to be opened.

1. To open a new document select the File|New command. A window appears asking for a file name and a location where the file should be stored. Enter the Continues Beam into the Structure name field and keep current settings for the Location field. Close the window with the OK button.

2. A simple wizard is used to collect some basic information about the structure. The first page asks about units that we use to describe the physical quantities of the structure and about its approximate size. Since this example is defined in

metric units, we recommend you to keep default settings. Don't worry, you will learn how to work with the Imperial units later. Our structure is 12 meters wide and about 0.5 meters high. Figure 2.2 shows the wizard's first page. Press the Next button to move to the second page.
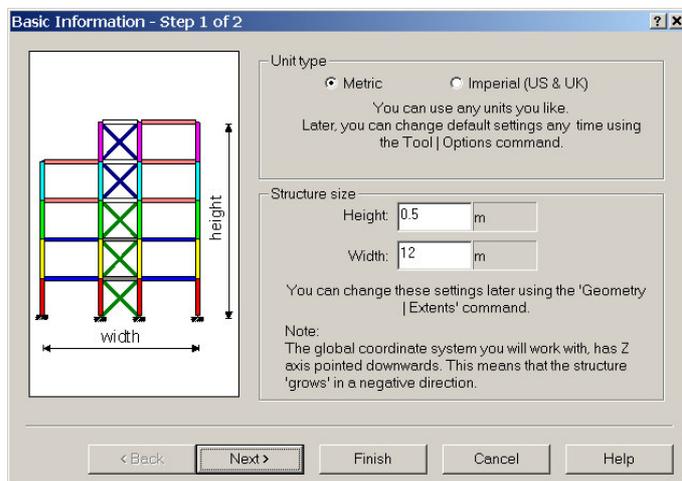

Figure 2.2: The initial size of structure.

3. The second page is used to describe the structure. The description is optional and AMSES Frame2D will use it as a comment. When working on real structures, we recommend you to take a few seconds and enter the structure description.

4. After the data was entered, close the wizard using the Finish button.

Comment: From the height and width given in the wizard the program reserves appropriate working space. The origin is set to the bottom left corner, which means that top right corner lies in positive x direction and in negative z direction of the world coordinate system. As you can see, the z-axis of the world coordinate system points downwards.
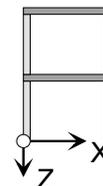
2.2.2   Defining the geometry

Maximize the main window and the working view. This will help you to track precise mouse movements. When the window is maximized, it should look like the figure 2.3.

The continuous beam is made of two elements. Both elements are six meters long. You start drawing the first element at the origin (0,0):

1. Move mouse pointer over the origin - coordinate (0,0). We can see current mouse pointer coordinate if we take a look at right side of the status bar. Press left mouse button to start drawing the first element.

2. Move mouse pointer to the right. Notice that a thin line is drawn from the origin to the pointer. Move the pointer to the coordinate (6,0).

3. Press left button and you will get the first element. The figure 2.4 presents the situation you should see on the screen – two unsupported nodes and an element.

Hold a mouse movement for a while to display a small tip window, which will provide useful information.

© 1999-2002 Ainet
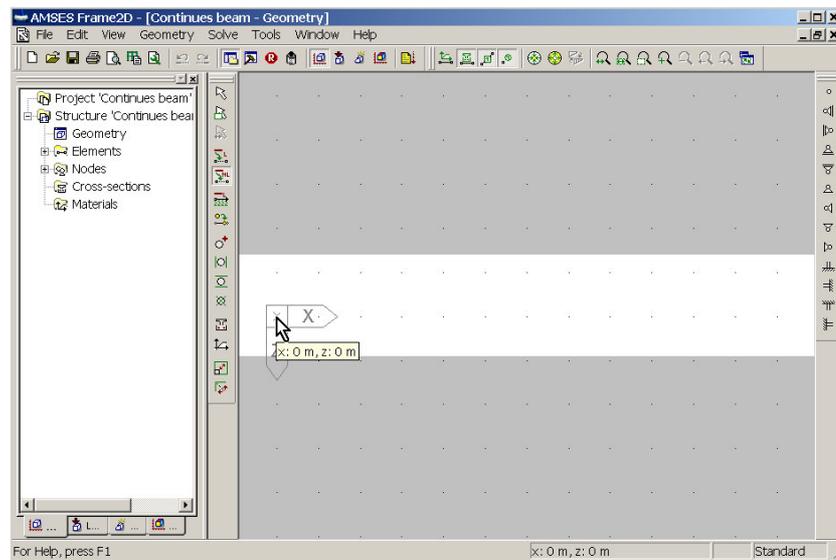
Figure 2.3: The geometry workspace window ready to take the mouse input.
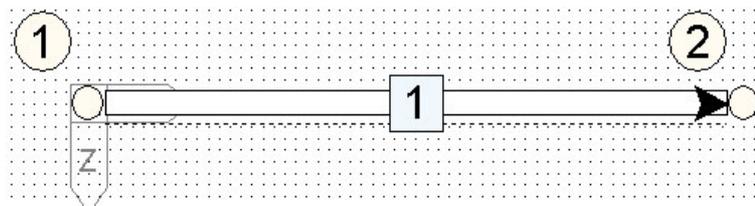
Figure 2.4: The first element is drawn.

We repeat the procedure to draw the second element:

1. Move pointer over node 2 and click it. Note: When the pointer is over a node, it slightly changes its shape – a small black dot appears. This indicates that the mouse click will not create a new node, but it will use the existing node instead.

2. Move the pointer to the right and locate coordinate (12,0).

3. When the pointer is in place, click left mouse button. Now you should see two elements as presented in the figure 2.5.
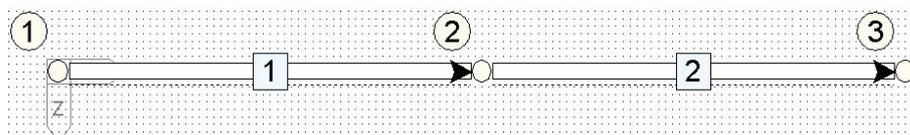
Figure 2.5: Both elements are in place.

Comment: While moving the mouse, do not keep left mouse button pressed. A line, a rectangle etc. are drawn in a slightly different way you may be used to. First, mouse button is clicked. Next, the pointer is positioned to the desired place. Finally mouse button is clicked again and this completes the operation. This approach differs from the approach that most windows programs use. AutoCAD uses similar approach, for example.

© 1999-2002 Ainet

Supports

Both elements are in place. Now we will support them by changing individual nodes into supports.

1. Locate the support toolbar on the right side and press on the ▲ icon, which represents a fixed pinned support.

2. Click on the node 1. Right after the click, the node changes its shape to the shape of the selected support.

3. Select the ▲ icon on the support toolbar – a movable pinned support.

4. Click on nodes 2 and 3. The screen should look as shown on the figure 2.6.
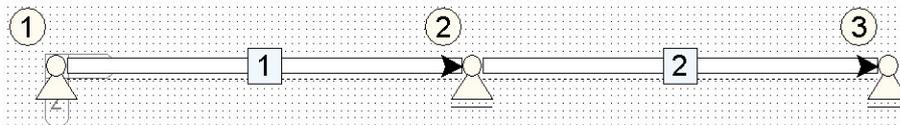
Figure 2.6: Supported continues beam.

Cross-section

In this step, we will define cross-section of both elements. When a new element is created it binds itself with a default cross-section. If the default cross-section does not exist, it automatically binds itself with a cross-section that has index 1, even if such cross-section does not exist. So far we did not define any cross-sections and for this reason, all elements point to a non-existing cross-section 1.

1. Select the Geometry|Cross-section command. This opens the cross-section editor window. Press the New ... button to create a new cross-section.

2. In a tabbed window, select the Rectangle tab and enter cross-section dimensions - depth 50 cm and breadth 30 cm, as shown on the figure 2.7.
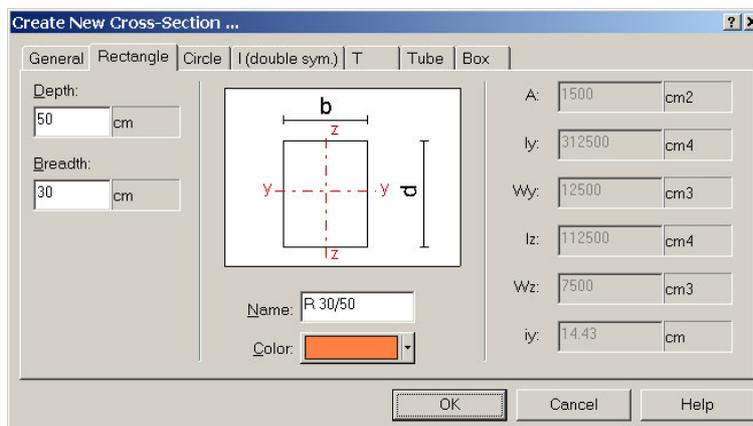
Figure 2.7: Entering data for a new cross-section.

© 1999-2002 Ainet

3. The cross-section should be named, too. Any name can be used. However, it is wise to use a meaningful name like R 50/30. A unique cross-section color should be also selected.

4. After the cross-section was defined, close the window with the OK button.

5. We are back in the cross-section editor window. Notice that the new cross-section R 50/30 appears in the cross-section list box. Index 1 was assigned to this cross-section, automatically. We have no control over indices in the current version of program.

6. After we have defined the cross-section, we MUST close the cross-section editor with the OK button. All modifications and new cross-section entries will be lost if we press Cancel.

Elements on the screen are painted by their cross-section color, by default. After the cross-section editor window was closed with the OK button, both elements change their color to the color of the new cross-section. Remember, both elements point to the cross-section 1. This is the reason why element colors match the color of cross-section 1.

Material

The material is defined in a similar manner as the cross-section:

1. Select the Geometry|Material command. This opens the material editor window. Press the New ... button to create a new material.

2. In a tabbed window select the Concrete tab and a page presented on the figure 2.8 appears.

Figure 2.8: Entering data for the new material. In most cases it is enough to select one from the database.

3. Selection from the database is handy. Choose the building codes first - Eurocode 2 and then browse for the C 30/37. Additionally, select a material color that uniquely identifies the material.

4. Our material is defined. Close the window by pressing OK.

5. We are brought back to the material editor window. Notice that the new material C 30/37 appears in the material list box. Index 1 was assigned to this material, automatically.

6. After the material was defined, we must close the editor pressing the OK button. All modifications and the new material will be lost if we press Cancel.

Now we are back to the geometry window. This time the elements on the screen remain unchanged. If we want to see elements painted in their material colors, we must issue the View|Show|Material command or press the ⤴ button on the geometry toolbar.

Press the ⤴ button to see elements painted in material colors.

Cross-section and material are defined and we can move forward to define actions, loads and load cases.

2.2.3   Actions and load cases

In order to define a load, we must invoke the Loads workspace first. This can be done in one of the three ways:

- By pressing the 🔧 button on the standard toolbar,

- by selecting the 🔧 tab under the three in the left pane,

- by selecting the View|Workspace|Loads command.

We notice significant change on the screen. Elements have disappeared, menus, toolbars and three in the left pane have changed. This is the default load workspace layout. In next step we will define actions and load cases.

Actions and load cases are defined in a special spreadsheet like window.

1. Double-click the Actions/Load Cases Manager line, which is located at the top of the tree on the left, or select the line and press Alt + Enter key. This opens the spreadsheet window.

2. Enter action names, load case names and load factors as defined at the beginning of this chapter, table 2.1. When we are finished, our spreadsheet should look like one on the figure 2.9.

	Limite state	Fi creep/Skew	G	Q
ULT. p&t G + Q (ult)	Ultimate P&T		1.350	1.500
G + Q (displ.)	Serviceability	2.200	1.000	1.000

Figure 2.9: The spreadsheet window after the entry has been completed.

3. Cells in the column Fi creep/Skew have special meaning. In this case they define the creep coefficient of the concrete. For the short term loads (ultimate load cases) the creep is not used, hence the cell is empty. However in long term loads (serviceability) we use value $\phi_c = 2.2$.

© 1999-2002 Ainet

Action load

Each action has a window of its own and we must open it before we can put any load on it. In our case, we will open the (G) window and define its load first.

1. Expand the Actions branch on the tree. You see a list of all actions defined in this structure.

2. Select item (G) and press Alt + Enter or double-click the item. A corresponding action window opens displaying a thin structure geometry ready to take some load.

Action (G) consists of uniform load distributed over both elements. Note that AMSES Frame2D treats the uniform load as a special kind of linear load. To define a load, a load type must be selected first.

1. Press the ⊞ icon on the toolbar or select the Add|Linear Force command to switch into linear load entry mode.

2. Click on the first element.

3. A dialog window appears asking about the intensity of the linear load. The figure 2.10 shows the dialog after load intensity has been entered. Enter 15 kN/m into the Fz field, set other fields to zero and name the load as Self weight.
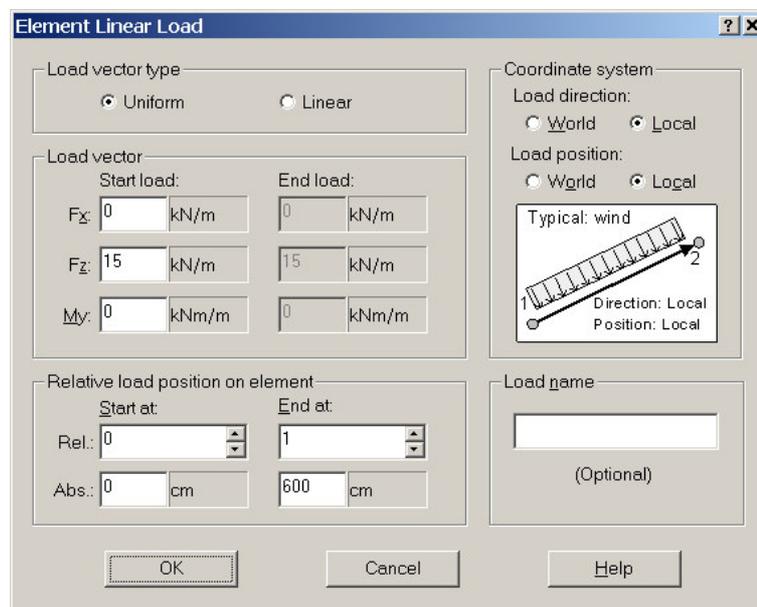
Figure 2.10: The linear load window. In this case, local coordinate system matches the world coordinate system, which makes the load direction and load position irrelevant.

4. Press OK to complete the operation. A load schema appears on the selected element.

The second element gets the same load, too. In principle, we could repeat the above procedure; however, there is a short-cut.

© 1999-2002 Ainet

1. Hold down the Shift key and click on the second element. The second element accepts the last entered linear load immediately – there is no need to fill the linear load dialog. 2.11.
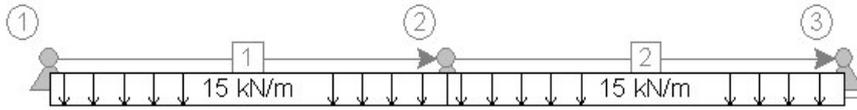
Figure 2.11: The continues beam and load of action G.

We have just defined the action (G). A similar procedure is used to define the load of action (Q):

1. Select the (Q) item in the tree and double-click it to open its action window.

2. Press the ⊞ icon to switch into linear load entry mode and then click the element one.

3. In the linear load window, enter 35 kN/m under Fz field and make sure that other fields are set to zero. Name the load as traffic load.

4. Close the dialog with OK. The element one has its linear load defined.

5. Hold down the Shift key and click the element two. Now, the second element has the same load as the first one. The screen should match the figure 2.12.
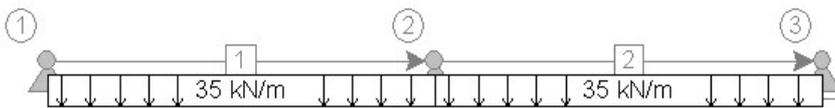
Figure 2.12: The continues beam and action load Q.

This completes the load definition.

2.2.4 Analysis

This is the easiest part. Simply press the 📄 icon on the standard toolbar or issue the Solve|Solve command. We can issue this command anytime. If anything goes wrong during the calculation, error messages are displayed in the message window, which is located at the bottom of the screen.

To browse the analysis results, we need to switch into results workspace. We can do this in three ways:

- Click the 🔧 tab below the three pane,

- click the ![icon] toolbar icon,

- select the View|Workspace|Results command.
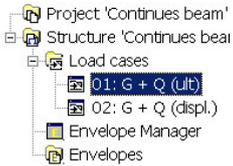
The contents of the screen changes after the switch. The load windows have disappeared, menus, toolbars and the three have changed. Now we see the results workspace layout. Results of each load case are presented in a separate window. To open a result window, we must expand the Load cases branch. Click on the plus sign and a list of load cases appears. When the list of load cases is visible, double-click the line named as G + Q (uls). This opens result window for the selected load case. If this is the first opened window, then bending moments are presented. The toolbar in the middle holds several icons, which can be described as shot-cuts to:

My⟳ ... display bending moments,

Fx→ ... display axial forces (tension is positive),

Fz↓ ... display shear forces,

u↘ ... display nodal displacements and reactions,

uL↓ ... display local element displacements,

φ⟳ ... display rotations,

⟱⁺ ... make diagrams larger comparing to the structure size,

⟱⁻ ... make diagrams smaller comparing to the structure size,

1,2 ... toggles result labels on/off. The toggle effects the printed output, too.

All these commands can be also reached via the Diagram menu.
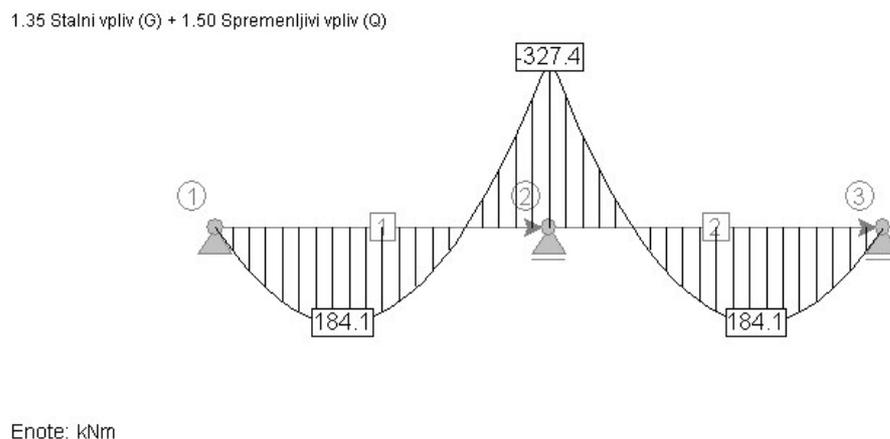
Figure 2.13: Results (bending moments) of the first load case.

Results of the second load case G + Q (sls) can be opened if you double-click appropriate line in the tree. To see desired internal force or displacements, simply select corresponding button on the middle toolbar.

Specifying number of calculation points

AMSES Frame2D calculates internal forces at the both element ends, at load application points and at equidistant points across the element. The number of equidistant points can be specified by the user. For the sake of design, it is wise to select maximum number of equidistant points regardless to increased computing time and memory requirements.

1. Issue the Tools|Options . . . command.

2. Select the Analysis tab.

3. Inside the Element results frame enter 19, which equals to 20 equidistant intervals across the element.

4. Close the window pressing OK.

The program remembers this setting and it will be applied for all subsequent cases.

2.2.5   Element design

Analysis results seem logical and we can move forward to element design. AMSES Frame2D does not really design elements but it checks if they are capable bear the load. The principle here to *specify and verify*. Some properties are specified first and its verification follows. We find such approach very useful. Namely, an engineer develops a special feeling for realistic results. The engineer is still in charge here. It is he or she which decides what properties a cross-section should have, not a computer program.

In our case, we will choose number of reinforcement bars and its disposition. Once this is done, the computer program will tell whether our selection fulfils the building code requirements.

The reinforcement is defined on element-per-element basis in the design workspace. We invoke the design workspace by one of the following actions:

- Click on the tab [icon] below the tree on the left,

- click the icon [icon] in the main toolbar,

- select the View|Workspace|Design workspace command.

We see similar workspace changes as in previous cases. Windows have disappeared and the tree has changed. In our case, both elements are made of (reinforced) concrete material. Hence, they are located in the Reinforced Concrete folder. If we expand the folder, we see both element names. Double-click on the [01] Element item opens the reinforced concrete element design window, shown on the figure 2.14.

The window is split into several sections. We are already familiar with the tree on the left. We use it to select individual elements. AMSES Frame2D enables us to design only certain cross-section shape and material combinations. Element that can't be designed are put into special Not designed folder. Currently, we can design rectangles
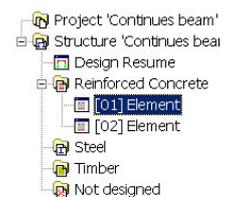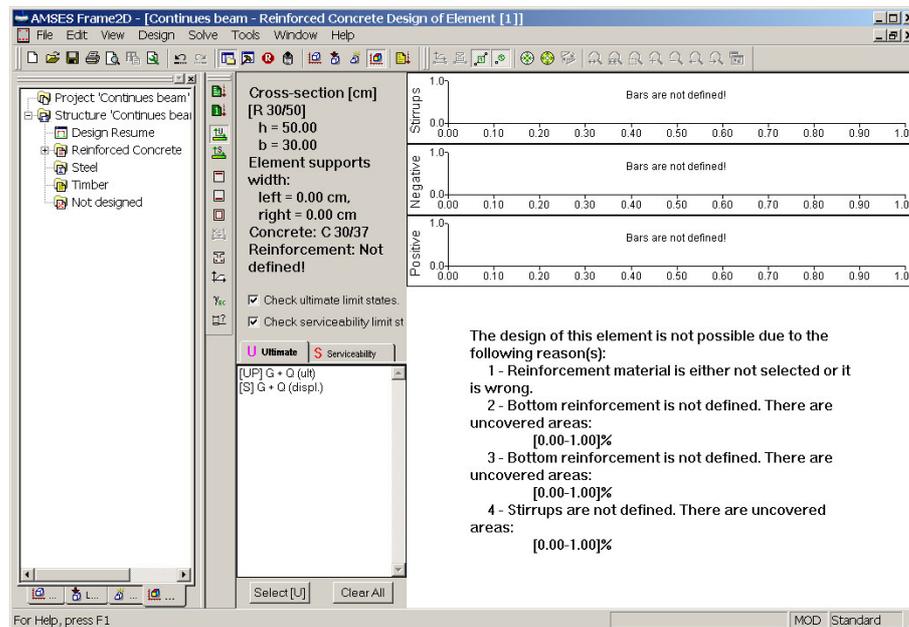
Figure 2.14: Initial look of the reinforced concrete design window.

and T shaped cross-sections made of reinforced concrete and tubes, boxes and I shaped cross-sections made of steel.

Let's focus back to the figure 2.14. The window is quite empty and the program has reported that it can't check the ultimate and serviceability states due to missing information. We must provide information about the reinforcement material, longitudinal bars disposition (positive and negative bars) and stirrups.

We will also need the support width at both element ends. Specifying the support width can significantly reduce bending moments at the intermediate support. Namely the theoretical length of the element can be quite larger than clear element length. Eurocode 2 allows us to take this into account.

Support width

Theoretical element length used in analysis goes from the mid point of the left support to the mid point of the right support.[1] Since the clear length is smaller than theoretical length used in the analysis, the calculated internal forces are slightly higher than the real ones. By defining the support width we can compensate this shear forces and bending moments peaks at both element ends.

1. Issue the Design|Edit Support Width command, or press the ⊞? icon located in the middle toolbar.

2. A window appears, figure 2.15. Here we enter width of the element supports at both sides. Caution! This is not the width of the complete support, we enter just

---

[1] The term *support* is used here in its general sence. Besides the true support, this term also covers cases of connections, say beam-to-column connection. If a beam is under consideration, it is evident that the clear length of the beam is smaller than theoretical length. In this case, we could specify one half of the column depth as the beam support width.

the portion that belongs to the element. In our case this is one half of the actual support width – 20 cm for either side.

3. Close the window by pressing OK.
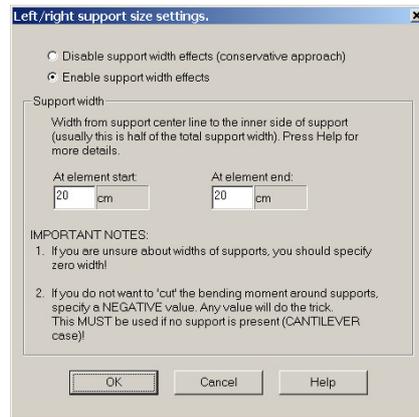
Figure 2.15: Support widths for our continues beam element. A support width reduces shear forces and bending moments.

Reinforcement material

The base cross-section material was already defined. Here we need to define an additional material used for reinforcement following given procedure:

1. Select the ⤧ in the middle toolbar or issue the Design|Edit & Select Materials ... command. A window opens where we select the base material and reinforcement material for our element. See figure 2.16.
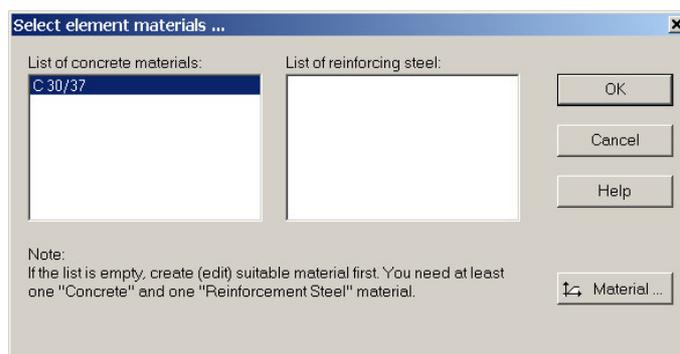
Figure 2.16: A window used to define concrete and reinforcement material. Reinforcement material is not known, yet.

2. The concrete material is already given. Only the reinforcement is missing. In order to select one, we must create it first. Press the ⤧ Material ... button and the material editor windows opens.

3. We need a new reinforcement material. Hence, press the New ... button.

4. Select the Reinforcement Steel tab.

5. Our material shall be RS 400. Its yield strength is 40 kN/cm2 or 400 MPa. Name the material as RS 400, which should be entered into the Material name field, figure 2.17. Specify the material color and press OK to close the window.
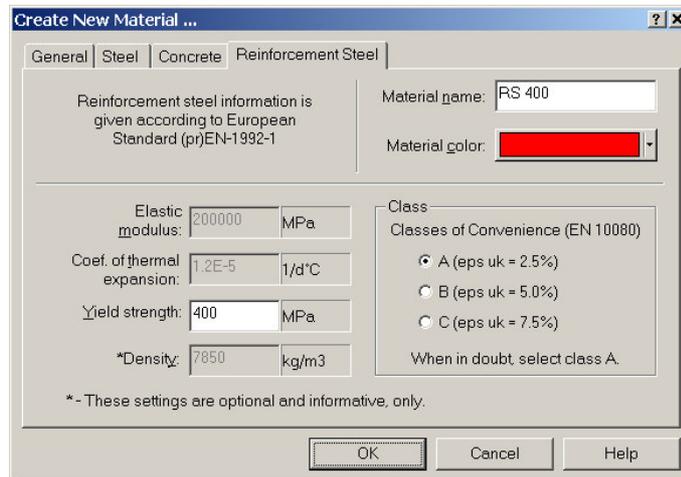
Figure 2.17: A material editor window.
.

6. We are back to the material editor. Close this one by pressing the OK button.

7. Now the reinforcement material is defined and appears on the reinforcement list. We must select (highlight) it. Being on the list it does not mean anything. A material must be highlighted in order to be selected, even if it is the only material in the list. The figure 2.18 reveals the correct selection for our element.

8. Close the window. The program has noticed the new reinforcement material. Now it complains for the longitudinal bars and stirrups only.
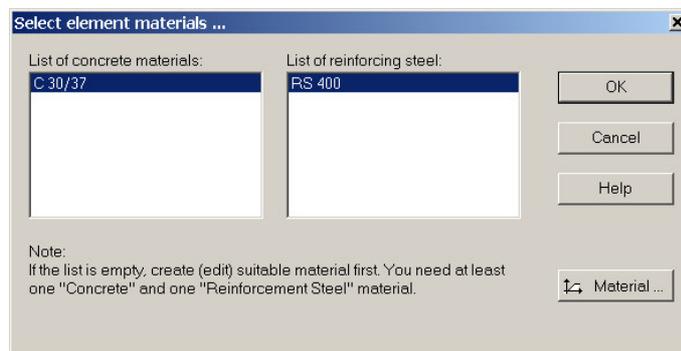
Figure 2.18: Reinforcement material is defined and selected (highlighted).

Longitudinal positive bars

Our next task is to define the quantity and disposition of the longitudinal bars and stirrups. The principle is the same in all cases.

For start we assume two $\phi 20$ mm bars along the whole element. The concrete cover[2] is 3 cm.

1. Select the Design|Positive Bars ... command. A window like shown in the figure 2.19 appears.
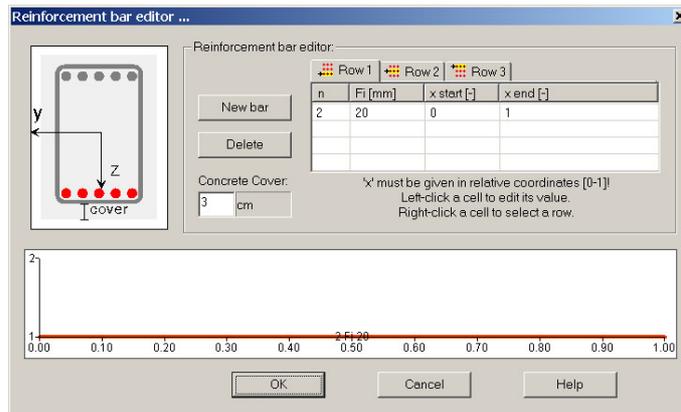
Figure 2.19: Correctly completed window for positive reinforcement.

2. In the spreadsheet we enter 2 under column n.

3. Under Fi we enter 20, under x start (bar start point in relative coordinates) we enter 0 and under x end (bar end point in relative coordinates) we enter 1.

4. For the Concrete Cover we specify 3 cm and the window should look like one on the figure 2.19.

5. Close the window by pressing OK.

Longitudinal negative bars

We will set the negative reinforcement in an almost identical way. Let's take two ϕ 20 mm bars for the start. The concrete cover is 2 cm.

1. Issue the Design|Negative Bars ... command.

2. In the spreadsheet, under letter n enter 2, under Fi enter 20, under x start enter 0 and under x end enter 1.

3. Into the Concrete Cover field enter 2 cm.

4. Press OK to close the window.

---

[2] Concrete cover is thickness of the concrete measured from the cross-section edge to the edge of the nearest stirrup or longitudinal bar.

Stirrups

Stirrups are defined in a similar way, too. Here we have an extra column in the spreadsheet – the distance between stirrup links.

Let's assume 2 times ϕ 10 mm stirrups. Around the support areas [0.0-0.2] and [0.8-1.0], we set them 7.5 cm apart. In the field area [0.2-0.8] we set them 15 cm apart.

1. Select the Design|Edit Stirrups . . . command.

2. Press the New bar button.

3. In the spreadsheet under column n enter 2, under Fi enter 10, under e enter 7.5 under x start enter 0 and under x end enter 0.2. This completes the [0.0-0.2] area.

4. Repeat the procedure for the field area. Press the New bar button. Under n enter 2, under Fi enter 10, under e enter 15, under x start enter 0.2 and under x end enter 0.8.

5. Only the third area is missing. Pres the New bar button. Under n enter 2, under Fi enter 10, under e enter 7.5, under x start enter 0.8 and under x end enter 1.0. The stirrup window is now complete and should look like the figure 2.20.
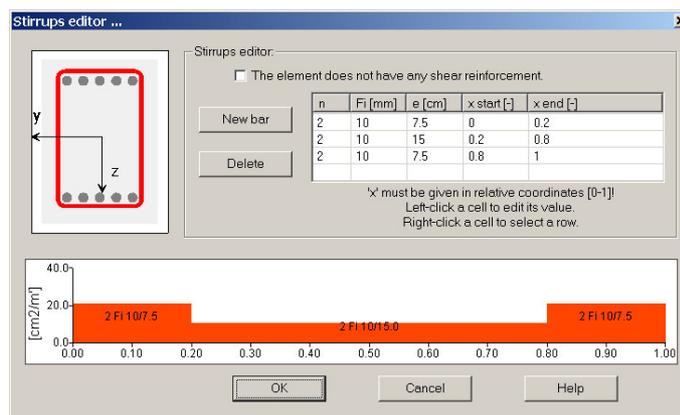
Figure 2.20: The look of the stirrup window after the stirrup information was entered.

6. Close the window by pressing OK.

Ultimate states calculation

Now almost everything is ready. In order to perform the ultimate limit states calculation, we must select which load cases should be used in design. Press the Select [U] button to select all ultimate limit state or simply select the [U] G + Q (uls) line from the list.

During the reinforcement material selection, the material list was changed and therefore AMSES Frame2D deleted the all analysis results. Hence the analysis must be repeated. Press the ⊟ icon or issue the Solve|Solve command. New analysis results are calculated. Now we issue the Design|This Element command or press the ⊟ button to verify the limit states. The program makes all possible design calculations and shows results in a compact form on the screen. The screen looks like the figure 2.21.
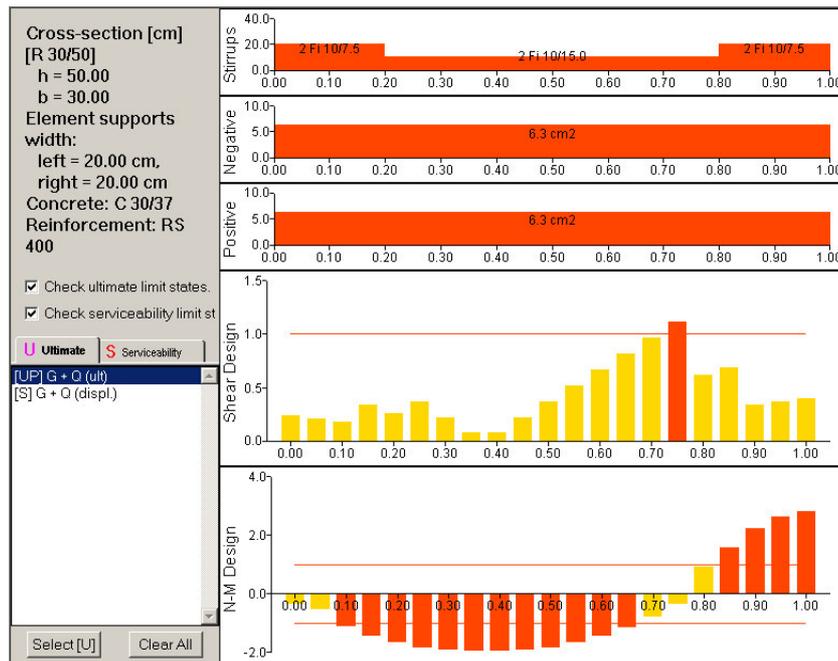
Figure 2.21: Once all required information is available, we can verify the design. Results are presented in a bar chart. All bars should be below 1. As we see, at some areas the cross-section capacity is exceeded.

Red colored bars mean nothing good. On this area the cross-section capacity is exceeded and we must either specify more reinforcement or make the cross-section larger.

In our case, we will fix the stirrups first. The chart shows that more stirrups are needed in the [0.7-0.8] area. In this area we have stirrups 15 cm apart. Let's reduce the field area to [0.2-0.7] and increase the end support area to [0.7-1.0]:

1. Issue the Design|Edit Stirrups . . . command – or double click the stirrups chart on the top.

2. Alter the second row in the spreadsheet. Under field x end enter 0.7. In the third row alter the x start field. Enter 0.7.

3. Close the window.

The stirrup situation is now fine. Still, we must fix the longitudinal bars. Let's deal with the positive reinforcement first. We will add an additional bar along the complete element and two more bars ϕ 20 mm over the [0.1-0.7] area.

1. Select the Design|Edit Positive Bars . . . command.

2. In the first row change the n from 2 to 3.

3. Press the New bar button and enter the following data into the new row: 2 under n, 20 under Fi, 0.1 under x start and 0.7 under x end.

4. Close the window.

© 1999-2002 Ainet

The positive reinforcement seem to be OK now. We still need to fix the negative one. To put it into the order we will add three bars over the [0.8, 1.0] area and one more bar over the [0.9, 1.0] area.

1. Open the negative reinforcement editor – Design|Edit Negative Bars . . . .

2. Add new row – New bar.

3. Enter: 3 under n, 20 under Fi, 0.8 under x start and 1.0 under x end.

4. Add new row – gumb New bar.

5. Enter: 1 under n, 20 under Fi, 0.9 under x start and 1.0 under x end.

6. Close the window by pressing the OK button.

Finally, the limit ultimate states requirements are fulfilled.

Serviceability limit states

At the end we must verify the serviceability limit states. At this stage of program development, we check the crack width only.

Let's switch to the serviceability limit states checks. Press the S Serviceability tab or press the icon.

In the list of load cases, select all load cases that needs to be verified against crack widths. In most cases, only serviceability load cases are verified. We can select them one by one or all of them by pressing the Select [S] button.

Once load cases are selected, a bar chart appears on the screen, figure 2.22. Bars represents width of the cracks. Since the criteria whether a crack is within limits or not changes from case to case, the program does not do any judgement here. It merely shows the crack width and allows designer to decide.
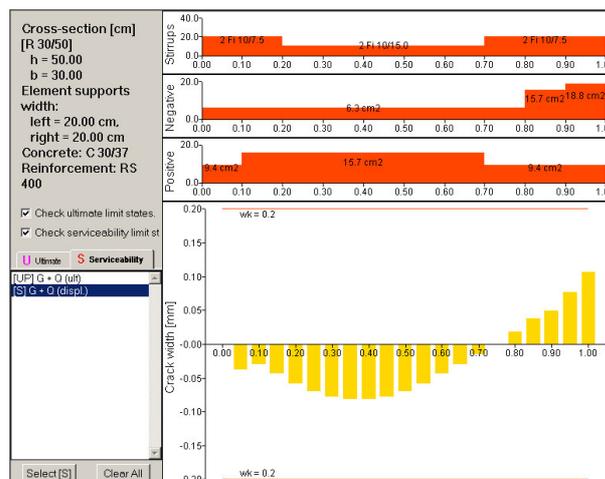
Figure 2.22: Crack width bar chart. The chart shows that no crack is wider than 0.1 mm.

© 1999-2002 Ainet

Design of the second element

The design of the second element is almost identical. There is not need to describe it here. We believe that you are able to do it yourself and we leave this as an exercise.

2.2.6 Printout

We came almost to the end. All we have to do is to print the results. Before we do the actual printout, we can preview the output on the screen, by clicking on the 🔍 icon or by selecting the File|Print Preview command.

Press the 🔍 and list the pages with PgUp and PgDn keys. You should see a screen like one on the figure 2.23.
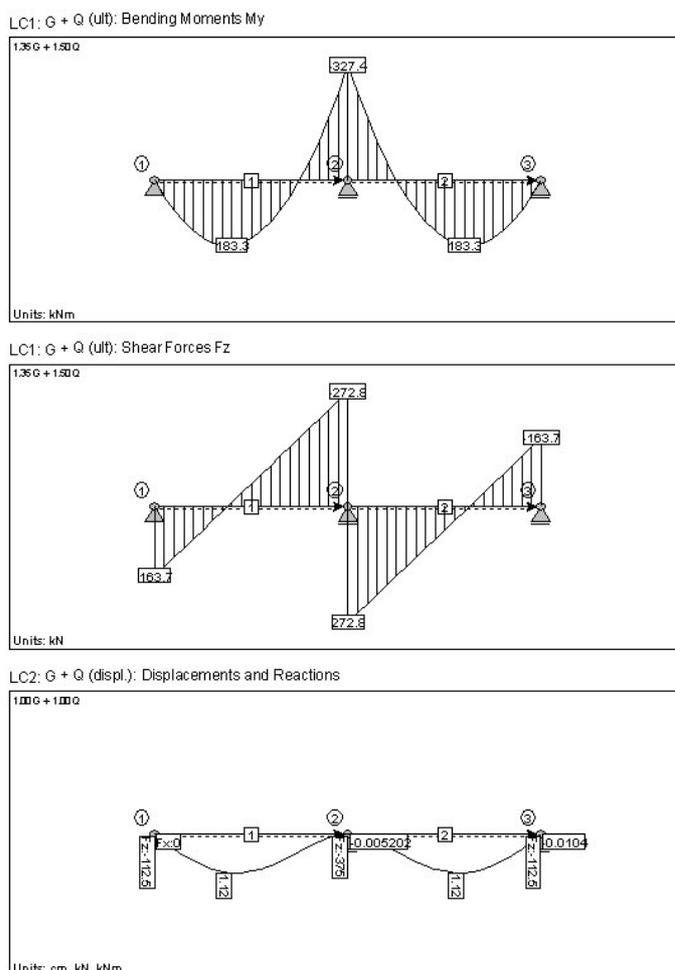
Figure 2.23: Predogled rezultatov na zaslonu.

In order to print the results, press the 🖨 icon or issue the File|Print command. Before printing the program asks which pages to print. In our case, keep the default (all pages) and press the OK button.

Output contents

When we are working on large cases, we get a significant amount of the printout. To reduce the printout, we use the Tools|Output Settings command. This command opens a special tabbed window, where we are able to define what shall be printed.

The most important are the Results and Geometry & Load tabs. We should exclude all tables and figures that are not needed. After the change it is wise to restart the print preview. This will tell us if the new setting were correct.

Comment: Some printer drivers do not process non-horizontal text (e.g. vertical reactions) properly. This error can be avoided by selecting the Tools|Output Settings command. On the General page, you will find the Check this box if ... checkbox. Check the box and close the window by pressing OK to correct this error. The printer output should work properly. However the print preview will be wrong.

2.2.7   Short conclusion

Your first structure has been successfully analyzed and designed. We also printed the results. You can handle other structures in a similar manner. Before you start using AMSES Frame2D on regular basis, we recommend you to read the rest of the manuals. There you will find many details that will enable you to use the program more efficiently.

The next chapter covers similar topic as this one, but deals with steel elements forming a simple frame.

© 1999-2002 Ainet

CHAPTER 3

An example of steel a frame and a beam on the elastic ground

The previous example has revealed only a few features of AMSES Frame2D. Another example was prepared to show more features and to make you familiar with the steel members design. A simple three element frame was constructed. The frame foundation is a RC-beam on elastic ground. During the analysis and design we will meet most of the essential program features:

- simple and efficient handling of geometric imperfection,

- using a precise non-linear finite element,

- beam on elastic ground – tension zone is excluded automatically,

- design of steel elements according to Eurocode 3.

3.1 About the structure

3.1.1 Geometry

The structure geometry is presented by figure 3.1. It consist of two columns rigidly connected by a beam. Both columns have pin connections on RC-beam. Precise dimensions of elements are not know at this point. Initially we can assume that columns are made of HE 240 A hot rolled profiles, while the beam is made of HE 360 A. Material S235 is assumed for all steel elements. The RC beam is of a T-shaped cross-section (depth = 60, breadth = 80, flange = 25, web = 40) and it is made of concrete C30/37. The wide part of the beam lies on the ground. The reinforcement grade is S400 ($f_y = 40$ kN/cm^2).

The subgrade characteristics are given by the subgrade reaction coefficient 0.015 kN/cm^3. The geometric imperfection angle is assumed to be $\phi = \frac{1}{260}$.

Beam has lateral and torsional support at both ends and one additional support at the middle. Both columns have lateral-torsional support at both ends only.

Nonlinear elements will be used to model the steel frame. Hence, buckling coefficients of individual elements do not need to be specially considered and we can assume to

take $k_y = 1.0$ for all elements. This assumption includes local material and geometrical imperfections, while the global imperfection is already taken into account during the analysis – due to initial imperfection and non-linear elements.
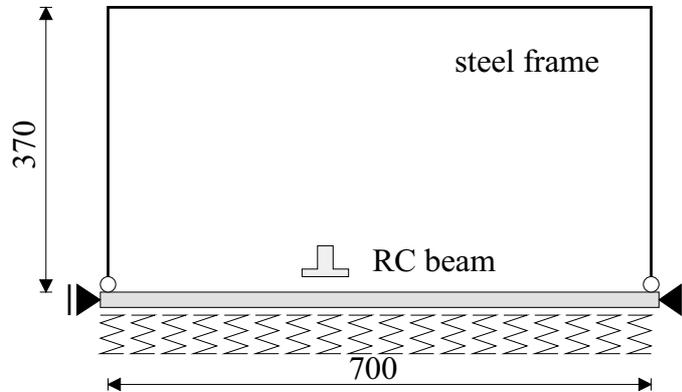
Figure 3.1: Mathematical model – geometry of the steel frame.

3.1.2   Actions – load groups

The steel frame is subject of several actions, which are listed below:

- (G) permanent and self weight action,

- (Q) variable action,

- (S) snow action,

- (W) wind action.

In next few steps we will briefly analyse individual actions. Unfortunately, a detailed analysis is beyond the scope of this manual.[1]

(G) Permanent action and self weight (dead load)

Standard weights were assigned to individual elements. The beam takes the self weight and weight of the composite slab. The columns take the weight of the facade elements and the self weight while the concrete beam only takes the self weight. Based on this data we calculated loads shown on the figure 3.2.

(Q) Variable load

Live load acts on the top beam, only. Actually, a load acts on the foundation beam, but such load is favorable – it reduces bending moments in the RC-beam – and is therefore neglected. We assume that the beam takes the uniform load of 25 kN/m. Additionally, 10 kN horizontal concentrated force acts on the left node. The figure 3.3 illustrates the situation. Due to simple geometry, there is only one most unfavorable position of the live load.

---

[1] However, in the future we intend to issue a simple tool that will handle actions in the Eurocode.
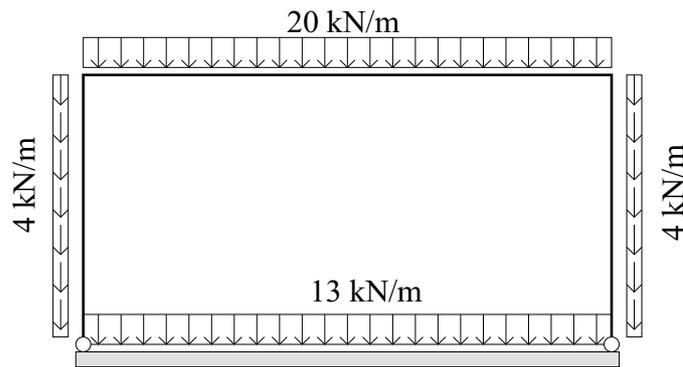
Figure 3.2: Distribution of the permanent action and self weight – (G)
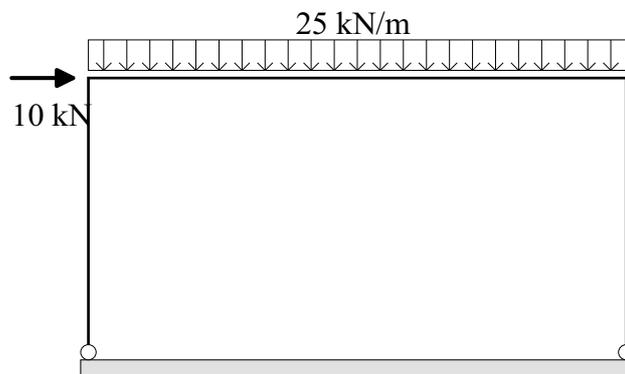
Figure 3.3: Distribution of the live load – (Q)

(S) Snow load

In general, two different snow loads needs to be assessed. In the first load (S1), the snow is evenly distributed over the beam. In the second load (S2), only half of the beam is covered with the snow and we take only half of te initial snow intensity. See figure 3.4.
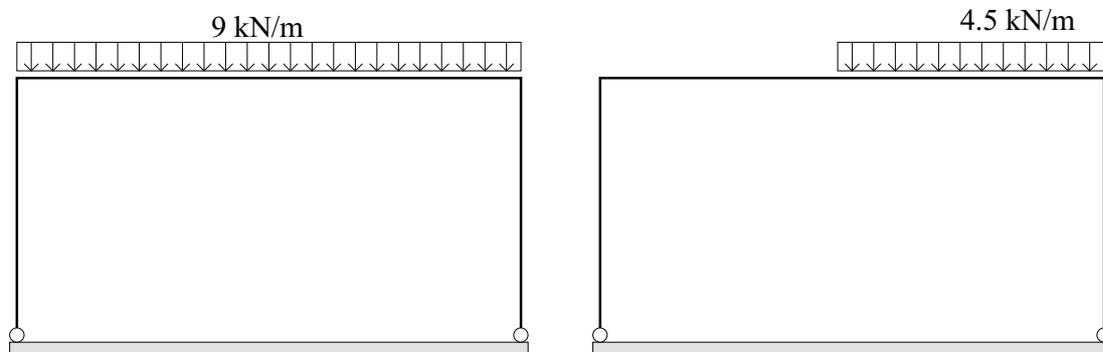
Figure 3.4: Both snow loads. The left one is labeled as S1 and the right one as S2.

Due to our simple geometry, we can realize that the second case S2 can never be mandatory and can be omitted. Hence, we will use only the first action – (S1 → S).

(W) Wind load

The wind can act from two different sides, hence two symmetric wind pressure distributions are assumed. The distribution for the wind blowing from the left side (towards the right side) is given on the figure 3.5. We get a symmetric picture, when the wind blows from the right.
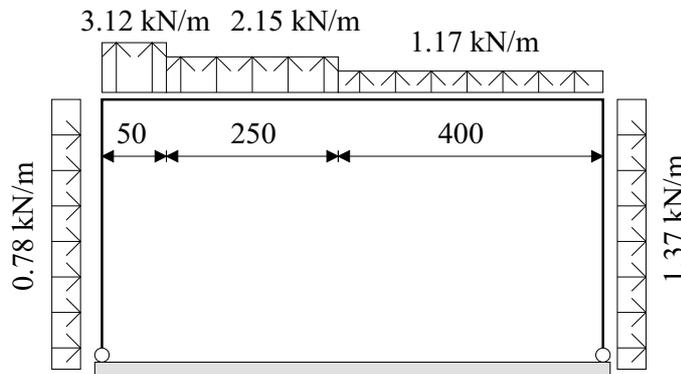
Figure 3.5: Wind load distribution load when the wind blows from the left (W).

All major actions are now described and we can focus on their combinations.

3.1.3 Load cases

In order to get a load case, we combine individual actions multiplied with partial safety factors. Load cases try to simulate some extreme conditions under which the structure can find itself during its lifetime (ultimate limit states). Additionally, load cases try to simulate rare, frequent and quasi-permanent conditions, which can be met under regular structure operation (serviceability limits states). Thus, each load case gets its individual set of partial safety factors (load factors).

Initial imperfection as a load case

AMSES Frame2D uses a special kind of load case – *skewness*. This load case allows the designer to set the initial imperfection of the structure. All load cases that follows are acting on skewed geometry until a new skewness load case is introduced. Hence, the sequence of the load cases is important.

In our case, we start with the skewness load case. We skew the structure to the right and appropriate ULS and SLS load cases follows. If the structure or the load is non-symmetric, then we should also skew the structure to the left using a new skewness load case and a new set of ULS and SLS load cases that follow. However, in our case we have symmetric situation and it is enough to consider load and skewness for one side only.

(a) Initial imperfection

Eurocodes demand that initial imperfection is taken into account during the analysis. The initial imperfection is based on a fact, that it is virtually impossible to make a structure perfectly vertical. It also includes effects that arise from residual stresses and

other deviations from the ideal material characteristics. Due to these imperfections, additional internal forces appear in the real structure and they need to be considered in the analysis too.

Eurocodes provide formulas based on number of stories and bays of a frame. The result of the formula is an equivalent imperfection angle for which the structure must be inclined (skewed) from the vertical. The structure must be skewed into direction which increases the internals forces. When in doubt, the structure should be skewed in both directions.

To follow ideas from the Eurocode, the program AMSES Frame2D defines special load case where we define the initial imperfection. This load case does not calculate any forces, but simply skews the structure for given angle. The same skewness is used in the load cases that follows until a new skew angle is specified.

Therefore, the skewness load case is the usually the first one. In our case, the structure is skewed for angle $\phi = \frac{1}{260}$ rightwards.

(b) Maximal vertical load

The structure is skewed and on this structure we put a combination of actions that gives maximal vertical load.

$$Vmax \leftarrow 1.35G + 1.50Q + 0.60 \cdot 1.50S$$

(c) Minimal vertical load

This is the opposite from the previous load case. The minimal vertical load is obtained by the following combination.

$$Vmin \leftarrow 1.00G + 1.50WL$$

(d) Maximal horizontal load – wind is not the leading action

A candidate for a maximal horizontal load is obtained if we take the maximal vertical load and add the wind action, where the wind is not the leading action.

$$Hmax \leftarrow 1.35G + 1.50Q + 0.60 \cdot 1.50S + 0.6 \cdot 1.50WL$$

(e) Wind from the left

Another candidate for the horizontal load is the variation of the previous load case, where the wind load is the leading action.

$$Wind \leftarrow 1.35G + 1.50WL + 0.70 \cdot 1.50Q + 0.60 \cdot 1.50S$$

(f) Snow

This is another variation of the loads, where the snow is the leading action. Wind is not included, since it acts favorable.

$$Snow \leftarrow 1.35G + 1.5S + 0.70 \cdot 1.50Q$$

Serviceability limit states combinations

The serviceability limit states are considered too. The steel elements will be checked for the stresses that should not exceed f_y and local deflections. The reinforced concrete foundation beam will be checked for crack widths. Having this in mind, we can define the following serviceability load cases.

(g) Maximal vertical load (SLS)

This case is needed to determine the beam deflection and RC beam cracks.

$$VmaxS \leftarrow 1.00G + 1.00Q + 0.6S$$

(h) Maximal horizontal load

This load gives us maximal sway displacements and contact pressures on the right.

$$HmaxS \leftarrow 1.00G + 1.0WL + 0.70Q + 0.6S1$$

Concrete creep and shrinkage

During the serviceability limit state analysis we can take into account reduced elastic modulus of the concrete due to creep and shrinkage effects. $E_{c\phi} = \frac{E_c}{1+\phi_c}$. In our case $\phi_c = 2.2$.

3.1.4   Load case table

All load cases from (a) to (h) are collected in a table form. All those cases are prepared for the structure which is skewed rightwards. The load case table is presented on table 3.1.

Almost identical table (spreadsheet) will be introduced when the problem will be solved using AMSES Frame2D, see the figure 3.9 on page 44.

Some abbreviations are used in the table. Their meaning is as follows:

- ULS (P/T) — ultimate limit state for the cases of persistent & transient type. This is the most frequently used type.

- ULS (S) — ultimate limit state of seismic type. This ULS type requires different material partial safety factors (lower) for the reinforced concrete materials. Hence, cross-sections have larger resistance.

- SLS — serviceability limit state. For the time being, we do not distinguish between different kinds of SLS.

By defining the table, we have all the information needed to perform the calculation using AMSES Frame2D.

© 1999-2002 Ainet

Load case	Type	ϕ_c	G	Q	S	W
(a) Skew rightwards	Angle $\phi = -\frac{1}{260}$					
(b) V max	ULS (P/T)	–	1.35	1.50	0.90	–
(c) V min	ULS (P/T)	–	1.00	–	–	1.50
(d) H max	ULS (P/T)	–	1.35	1.50	0.90	0.90
(e) Wind	ULS (P/T)	–	1.35	1.05	0.90	1.50
(f) Snow	MSN (P/T)	–	1.35	1.05	1.50	–
(g) V max S	SLS	2.20	1.00	1.00	0.60	–
(h) H max S	SLS	2.30	1.00	0.70	0.60	1.00

Table 3.1: Load cases, trier types and partial safety factors.

3.2   **AMSES Frame2D** solution

Let's take a look at the AMSES Frame2D solution. We recommend you to run the program and follow the step by step instructions from the manual. Most of the procedures that were introduced in our first example are also used here. They are repeated here in a more condensed form. The final solution can be found in the file:

..\ProgramFiles\AMSES\Frame2D\Examples\SteelFrame.amp.

3.2.1   Opening a new document – structure

Before entering the geometry of the structure, a new document – structure needs to be open and some basic information needs to be given.

1. Start AMSES Frame2D and open a new document – structure by issuing the File|New command. A window opens where we define the name of the structure and the folder where the file will be stored. Press the OK button.

2. Select metric units and enter the structure extends in order to reserve appropriate drawing area. Our structure is about 4 m tall and 7 m wide. Proceed with the Next button.

3. Enter relevant information about the structure (this is optional) and close the window by pressing the Finish button.

Once the process is completed, the geometry workspace appears and the geometry window is opened. It is wise to maximize it to make the drawing easier.

3.2.2   Structure geometry

The figure 3.1 shows the frame geometry. This is a simple orthogonal frame. The upper part consists of three steel elements, while the bottom part consist of a reinforced concrete beam that lies on the elastic ground.

It is important, to select *non-linear elements* to model the upper part of the structure (steel elements), while the foundation beam needs to be modeled using *linear elements*. Foundation beam lies on the elastic ground and AMSES Frame2D allows such simulation for linear elements, only. See chapter 7 on page 89 for more details.

Element on the elastic ground must be modeled using linear finite elements.

Once the geometry is set, the situation on the screen should look like the figure 3.7.

Steel frame

Let's draw the left column first.

1. Make sure that the ⬛ icon is currently active. Steel element should be modeled using non-linear finite elements.

2. Move the cursor to the coordinate $(x,z) = (0,0)$ m and click left mouse button.

3. Move the cursor upwards and click on the $(x,z) = (0,-3.7)$ m coordinate to finish the element.

Repeat the above procedure for the right column. The only difference is that the left column is at $x=7$ m. Once the second column is in place, the beam can be drawn. The beam connects nodes 2 and 4.

1. Click on the node 2.

2. Move pointer to the node 4 and click again.

The steel frame is now completed. If you find the elements, nodes and labels too large, you can make them smaller by pressing the ⬛ icon a few times.

Beam on elastic ground

There are two issues about the beam on elastic ground that need to be mentioned.

- The beam must be modeled using linear elements.

- The local coordinate system of the beam is very important. The T-shaped cross-section has its web always on the positive side and the flange on the negative side. In order to make the flange lying on the ground, the element must be drawn from right to left.

Therefore:

1. Select the linear element – click on the ⬛ icon.

2. Move the pointer over the node 3 and click.

3. Move the pointer over the node 1 and click once more to complete the beam. The element must have green frame, which indicates, that it is a linear element.

4. We must assign springs (bi-linear) to the element. To do this, select the ⬛ icon from the toolbar.

5. Click on the beam and a window opens, figure 3.6.

6. Enter the 0.015 kN/cm^3 value for the subgrade coefficient and select the negative side, as shown on the figure.

7. Press OK to close the window.

© 1999-2002 Ainet
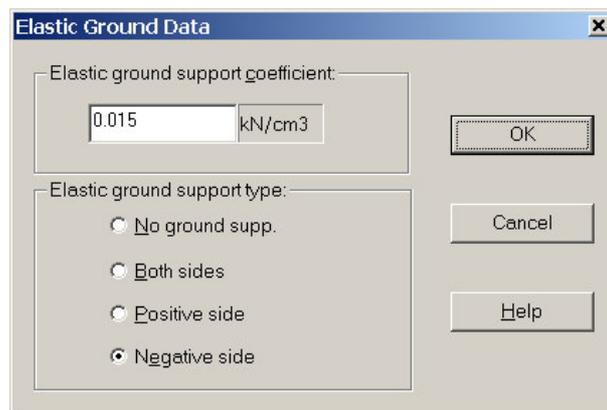
Figure 3.6: A window where subgrade coefficient and its position are specified.

Supports

If we look at the structure as a rigid body, we realize that the structure is not stable. The elastic springs supports the structure at the bottom, but they do not prevent the movement in the horizontal direction. Hence, additional supports are needed, as shown on the figure 3.1, page 32.

1. Select the icon from the toolbar on the right and click the node 1. This makes the structure stable.

2. Nevertheless select the icon and click the node 3 to make the both sides supported.

Element releases

Currently, both columns are rigidly connected to the foundation beam. This is not in correspondence with the figure 3.1 on page 32. We need to release rotations of both columns at the foundation beam connection. The following procedure will release the columns:

1. Click on the to turn on the rotation (bending moment) release.

2. Click on the low part of the column 1 (near connection to the foundation beam). A special symbol that indicates the rotational release appears.

3. Click on the low part of the element 2 to release the rotations of the second column, too.

Cross-sections

We have three kind of cross-sections: (a) both columns have the same dimensions, (b) steel beam has a cross-section of its own, as well as the foundation beam (c). Let's define them here and modify them later, if necessary.

1. Issue the Geometry|Cross-Section . . . command. A window opens.

2. Let us define the column cross-section first. Press the New button and select the I (double sym.) tab.

3. Check the Standard sections box and a list of standard hot-rolled sections appears.

4. Select the HE (EU) wide-flange from the top list and the HE 220 A from the bottom list.

5. The cross-section needs a color. Pick up the red one.

6. Close the window by pressing the OK button.

7. Now we need to define the beam cross-section. Press the New button and select the I (double sym.) tab.

8. Check the Standard sections box and a list of standard hot-rolled sections appears.

9. Select the HE (EU) wide-flange from the top list and the HE 360 A from the bottom list.

10. Pick the green color for this cross-section (bream).

11. Close the window by pressing the OK button.

12. Finally we need the foundation beam cross-section. Press the New button and select the T tab.

13. Enter 60 cm into the Depth field, 80 cm into the Breadth field, 40 cm for the Web thickness and 25 cm for the Flange thickness.

14. Label the cross-section as T-beam and pick the yellow color for it.

15. Close the window by pressing the OK button. This puts us back to the cross-section window. Click OK once more to complete the cross-section input.

All elements on the screen are colored in red (the color of columns). We need to change the cross-section of both beams:

1. Move the cursor over the top (steel) beam and press the *right* mouse button to display the pop-up menu.

2. Select the Properties menu item and the element property window appears.

3. Select the Cross-section tab and browse for the HE 360 A in the list.

4. Click on the Apply button, to make the new selection active. The beam on the screen (if it is visible) turns green.

5. Click on the pin icon to make the property window *pinned* on the screen.

6. Click on the bottom (foundation) beam and repeat the procedure. Here we must select the T-beam cross-section. Do not forget to hit the apply button to accept the changes.

7. Now we can close the property window.
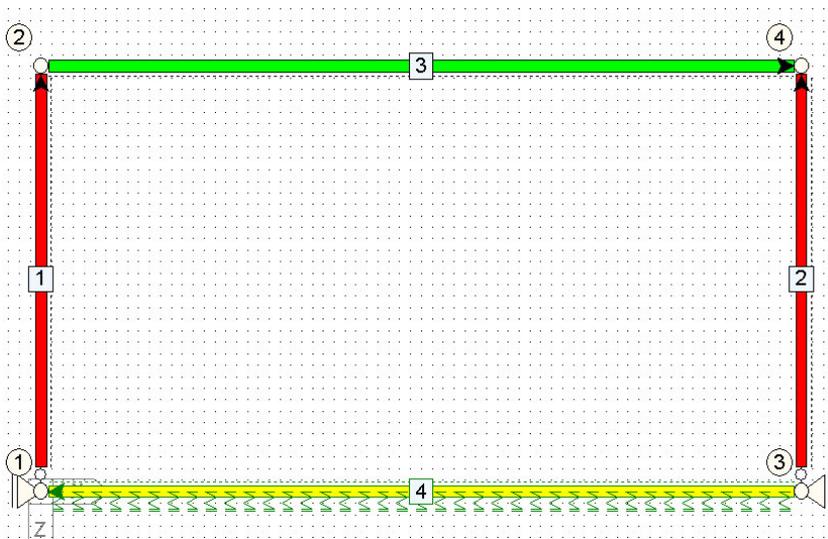
Figure 3.7: Situation on the screen after the geometry and cross-section were de-
fined.

Materials

This structure uses thee different materials: a structural steel, a concrete and a rein-
forcement steel. We need to define all three here.

The structural steel of the steel elements has S 235 grade, the foundation beam is
made of C30/37 grade concrete and of S400 reinforcement. For the sake of analysis,
the reinforcement is not really needed. However, it is needed later during the design
stage and it is wise to define it at this point as well.

1. Select the Geometry|Material . . . command. A window opens.

2. Press the New button and select the Steel tab to declare a structural steel mate-
   rial.

3. Select the Eurocode 3 [2] from the Code list and material S 235 from the material
   list.

4. Pick the dark red color for the selected steel.

5. Press OK to close the window.

6. In order to define the concrete, press the New button again and select the Con-
   crete tab.

7. Pick the Eurocode 2 from the top list and C 30/37 from the bottom.

8. Select light gray color and close the window.

9. The reinforcement material is still missing. Hit the New button once more.

10. Select the Reinforcement Steel tab. Note: Steel and Reinforcement steel are com-
    pletely different material types and they must never be interchanged.

11. Enter 40 kN/cm^2 or 400 MPa into the Yield strength field and RS 400 into the
    Material name field. Pick pink color for the reinforcement.

12. Press OK to close the window.

13. Close the material main window to complete the definition of materials. Press the OK button.

After the material definition, the screen does not change. The element colors still refer to the cross-sections. To alter this situation, press the 🔺 icon on the top toolbar or issue the View|Show|Material command. Now the elements are viewed in their material colors. The bottom beam color is wrong. It is dark red (steel) while it should be light gray (concrete).

1. Right click on the foundation beam element and select the Properties command.

2. Select the Material tab from the element properties window.

3. Display the material list and select the C 30/37 material for the main material and RS 400 for the reinforcement.

4. Do not forget to hit the Apply button to put the changes into effect. The beam should turn light gray.

3.2.3   Load – actions and load cases

To describe the load of the structure, we must start with the action and load case definition first. Both were defined in the initial section of the chapter and they were summarized in the table 3.1 on page 37.

AMSES Frame2D uses a similar table to define actions and load cases. The final result for our case is presented on the figure 3.9 on page 44.

1. Switch AMSES Frame2D into the load workspace. This can be done by issuing the View|Workspace|Loads command.

2. Double-click the Action/Load case Manager line on the three and a spreadsheet appears. The topmost row defines actions, while the leftmost column defines load cases.

3. Rename the Action 1 cell into G and the Action 2 cell into Q.

4. In oder to enter the next two actions, the spreadsheet needs to be expanded. Press the 🔲 icon twice to add two more columns. The same effect can be obtained using the Load Manager|Add|Action command.

5. Rename the new columns into S and W.

Actions are now defined. Each action gets its own window, where we define its load. At this moment we could start putting loads onto the frame. Nevertheless, let's define the load cases and load factors first. All we have to do is to copy numbers from the table 3.1 into the spreadsheet.

1. Rename the Load case 1 cell into Skew rightwards. The first load case defines the skewness of the structure. Such skewness is then used in subsequent load cases until a new skewness is defined. We are going to skew the structure for $\phi = -\frac{1}{260}$. A negative value must be used, since the skew is a consequence of the clockwise rotation, which is a negative rotation.

2. Click on the Limit state cell and click the arrow to open the list box. Select the Skewness type. To bring selection into effect, click outside the cell. Once this was done, the cell turns gray.

3. Click on the gray cell to open the skewness window, figure 3.8. In our case we enter -260 into the Horizontal field for the angle, while we leave the axis location as it is.[2]

4. Confirm the entry by pressing OK.
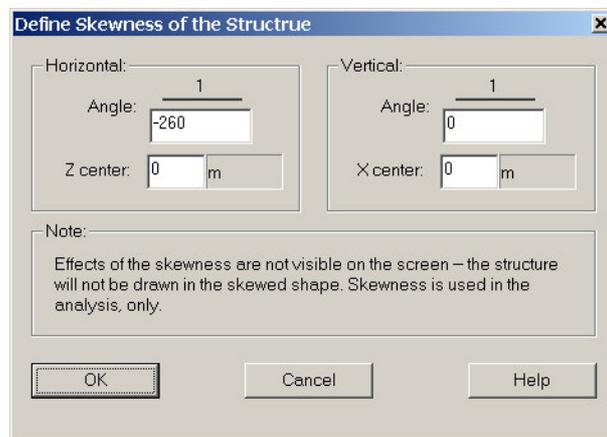
Figure 3.8: The skewness window. In reality we skew the structure by prescribing the angle of the skew.

5. Rename the second load case from Load case 2 into V max.

6. This is an ultimate load case. Hence, select the Ultimate P&T from the Limit State column.

7. The Fi creep/Skew cell shall remain empty.

8. Enter load factors that correspond from the table 3.1: G=1.35, Q=1.50, S=0.90, W=0.00.

9. Add new load case by issuing the Load Manager|Add|Load case command or by pressing the 🔲 icon.

10. Rename the new row into V min, select the Ultimate P&T limit state and enter corresponding factors into cells. See table 3.1.

---

[2] The skewness effect can be visualized. Issue the Solve|Solve command, which calculates internal forces and displacements. The same command also calculates the initial skewness. To see the results, switch to the results workspace (View|Workspace|Results) and double click the 01: Skew rightwards line. Finally, select the 🔲 icon to view the displacements.

© 1999-2002 Ainet

11. Repeat this procedure for all other load cases. The only thing we should take care about is that the V max S and H max S have different limit state. They are Serviceability load cases. This is important from the design point of view.

	Limite state	Fi creep/Skew	G	Q	S	W
Skew rightwards	Skewness	Z:1/-260				
V max	Ultimate P&T		1.350	1.500	0.900	
V min	Ultimate P&T		1.000			1.500
H max	Ultimate P&T		1.350	1.500	0.900	0.900
Wind	Ultimate P&T		1.350	1.050	0.900	1.500
Snow	Ultimate P&T		1.350	1.050	1.500	
V max S	Serviceability		1.000	1.000	0.600	
H max S	Serviceability		1.000	0.700	0.600	1.000

Figure 3.9: The look of the action/load-case spreadsheet after the entry was completed.

3.2.4 Load – actual load of individual actions

Actions have been already defined in the section 3.1.2, which start on page 32. We will load the frame elements and nodes according to the schemas presented in the section.

Action (G)

The load disposition for this action is presented on page 33, figure 3.2. To arrange the load in accordance with the figure, the following procedure should be followed:

1. Make sure that the tree on the left has the Action branch expanded. If this is not the case, click on the [+] in front of the branch.

2. Double click on the action denoted by G. This opens the load window for selected action.

According to the figure, the top beam gets uniform load of 20 kN/m.

1. Select the linear load icon ⊞ or issue the Add|Uniform Force command. Note: uniform load is just a special case of linear load.

2. Click on the top beam and linear load window appears. Enter 20 kN/m into the Fz field, select World for the Load direction and Local for the Load position.[3]

3. Confirm the entry by pressing OK.

Both columns are loaded with 4 kN/m. The procedure is almost identical here:

---

[3] We are dealing with the gravity load here. Since the gravity load always points downwards, the load direction refers to the global coordinate system. However, the intensity of the load depends on the beam length, hence the position of the load refers to the local coordinate system.

1. Click on the left column and enter 4 kN/m into the Fz field. The load direction and the load position settings are already in place (the self weight illustration shall be visible). Press OK to close the window.

2. The right column gets the same load. To copy the last load, hold down the SHIFT key and click on the right column.

Both columns are loaded now. Remember: You can use Edit|Undo to reverse the last command.

We still need to load the bottom beam. The beam is loaded with 13 kN/m. You can try this for yourself. When you are finished, your screen should look like the figure 3.10
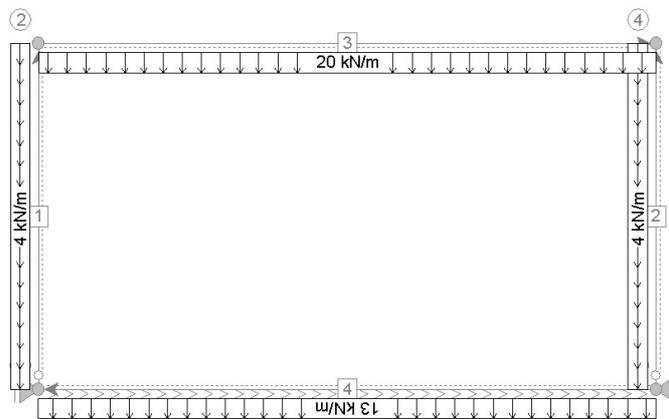
Figure 3.10: The load of the action (G)

Action (Q)

The live load was introduced on page 33, figure 3.3. It consists of one uniform load and one concentrated force. Before the load can be specified, action Q window must be opened.

1. Double-click the Q line on the three. A new window opens, which corresponds to the action Q.

2. Select the ⊞ icon and click on the top beam.

3. The linear load window opens. Enter 25 kN/m into the Fz field and choose correct coordinate systems. (Our live load behaves as a self weight.)

4. Close the window.

5. Select the ↻ icon and click on the node 2.

6. The concentrated load window opens. Enter 10 kN into the Fx field and hit OK button to exit.

The live load (Q) is now completed.

Action (S)

Initially, two different snow situations were considered but we realized that in our case only the first one can be mandatory. See page 33, figure 3.4. Let's define the snow load.

1. Double-click the S line on the three. A new window opens, which corresponds to the action S.

2. Select the ⊞ icon and click on the top beam.

3. The linear load window opens. Enter 9 kN/m into the Fz field and choose correct coordinate systems – both, load direction and load position must be in *global* coordinate system.

4. Press OK to close the window.

Action (W)

Figure 3.5 on page 34 shows the distribution of wind that we must define for this action. Here we will take into account that the wind acts always perpendicular to an element. Hence both the load direction and load position refer to the *local* coordinate system.

1. Double-click the W line on the three. A new window opens, which corresponds to the action W.

2. Select the ⊞ icon and click on the top beam, where we must define three different loads. The linear load window opens.

3. Select *local* for load position and load direction. Enter -3.12 kN/m into the Fz field.

4. Since this load refers only to the left part of the beam, we must specify the load start and end position. This can be entered using relative or absolute coordinates. We will use absolute coordinates here. Enter 50 cm into the Abs row under the End at column.

5. Press OK to close the window.

6. Click on the top beam once again and enter the -2.15 kN/m into the Fz field.

7. This load starts at $x = 50$ cm and ends at $x = 300$ cm. Enter 50 cm into the Abs row under the Start at: column and enter 300 cm under the End at: column.

8. Press OK to close the window.

9. The remaining of the beam is loaded with 1.17 kN/m. Click on the beam and enter -1.17 kN/m into the Fz field.

10. The load starts at $x = 300$ cm and lasts up to the end. Enter the 300 cm into the Abs row below the Start at column. The end position is already in place.

11. Press OK to close the window.

Both columns are loaded too. The left column takes 0.78 kN/m while the right one takes 1.37 kN/m.

1. Click on the left column, enter 0.78 kN/m into the Fz field and press OK to close the window.

2. Now, click on the right column and enter 1.37 kN/m into the Fz field. When you press OK to close the window, the wind load is completed.

If everything went well, your screen should look like the figure 3.11. We have just defined the last action and we can move on to the analysis.
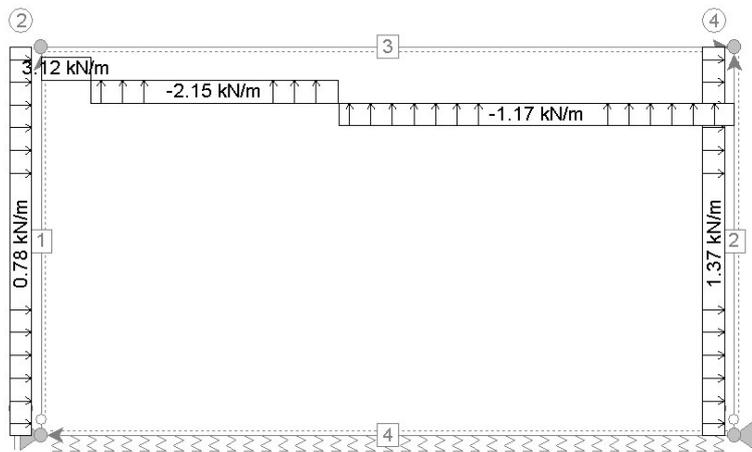
Figure 3.11: The load of the action (W) – wind load.

3.2.5   Analysis

The geometry and load of the structure are completely defined and we can calculate internal forces and displacements. To view them we must move to the *Results* workspace. Issue the View|Workspace|Results to move there.

The tree shows load cases that we have defined in the spreadsheet. Since the analysis has not been performed yet, all windows are empty. In order to perform the analysis issue the Solve|Solve command. After the command had completed the job we can browse results for each individual load case.

Envelopes

Sometimes we have to deal with a large number of load cases. In order to have one look over all load cases (or over a subset of load cases) we define an envelope for a particular quantity (an internal force, displacement, contact pressure). Usually, bending moment M_y is selected for the *leading* quantity. Any number of envelopes can be defined. Let us define two envelopes.

- Envelope of bending moments M_y. We will include all ultimate limit state load cases.

- Envelope of contact pressures. Here we will include only serviceability load cases.

Envelope of bending moments

An envelope is defined in a so called Envelope Manager.

1. To open the Envelope Manager window, find the corresponding line in the tree and double-click it. Note: Maybe you need to scroll down the tree to see the line. The window is shown on figure 3.12.
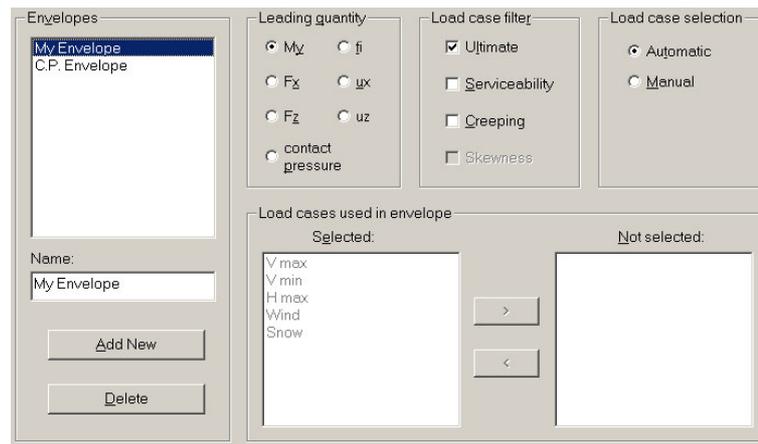
Figure 3.12: The Envelope Manager window. The window shows values and settings needed to define the envelope for bending moments.

2. Define a new envelope – press the Add New button.

3. Enter My Envelope into the Name field.

4. Select the leading quantity – My.

5. Specify which load cases shall be included in the envelope. In our case all Ultimate load cases are included, hence turn on the Ultimate checkbox.

6. It is also advisable to keep the automatic load case selection turned on. In this case, all ultimate limit states are automatically included into the envelope. If the Manual selection is in effect, we have to pick load cases one by one.

7. Bottom lists show which load cases are selected and which are not. Note: Sometimes it is necessary to click on the Automatic button even if it is already selected in order to activate the automatic selection.

Envelope of contact pressures

We still need to define an envelope of contact pressures. The procedure is almost identical.

1. Press Add New button to define a new envelope.

2. Enter C.P. Envelope into the Name field.

3. Select the contact pressure option for the leading quantity.

4. Check the Serviceability option and clear the Ultimate option in the load case filter.

5. Keep the automatic selection of load cases turned on.

Envelopes are now defined and we can look at the results. In order to browse the envelope results, we need to expand the Envelope branch on the tree. Once the branch is expanded, a list of envelope names appears. Double click an envelope name to open the results window and browse the results in the same way as you browsed the load cases. Figure 3.13 shows the results for contact pressures.
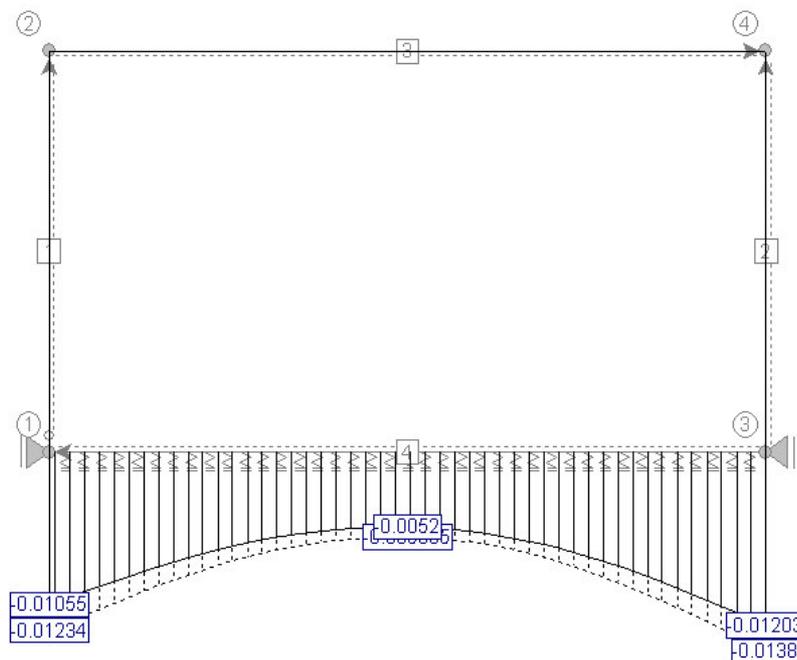
Figure 3.13: Envelope of contact pressures. Boxed numbers (pressures) are given in kN/m^2 units.

3.2.6 Design of steel elements

Once the structure is stable and the internal forces are calculated, we can design its elements. Note that envelopes we have just defined have nothing in common with the design. During the design each element is verified against all selected load case individually.

Before the design takes place, it is wise to define groups of elements that shall have the same cross-section properties. In our case, we have already done this – both columns have the same cross-section, the top beam has its own, as well as the bottom beam.

Top beam – element 3

Now it is time to perform design for each element. We will start with the top beam. The design takes place in the design workspace. Select the View|Workspace|Design command or press the ![icon] icon to switch into the workspace.

In the tree you can see several branches which hold individual elements. The Steel branch holds elements, which can be verified against rules and principles of Eurocode 3 (steel design), the Reinforced concrete branch holds elements verified against Eurocode 2, the Timber branch is always empty, while the Not designed branch holds elements that do not fit into any other branch and can't be designed by AMSES Frame2D.

Expand the Steel item and double click element 3. The situation on screen changes and it looks like the figure 3.14. Alternatively, double click the Design resume item. This opens a window where all elements are presented. Now double-click the top beam-element to open its design window.
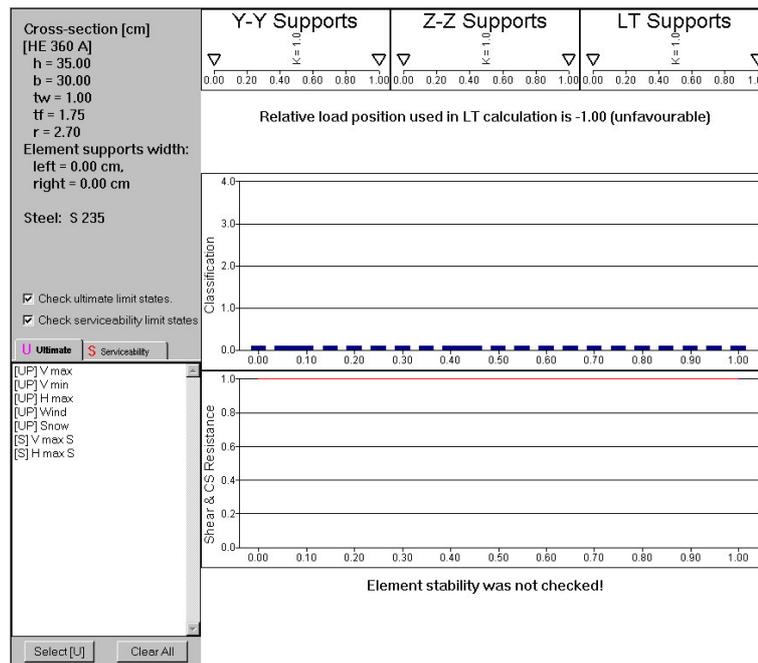
Figure 3.14: Initial look of the steel element design window.

Supports: The beam geometry has not been completely defied, yet. Namely, the beam has an additional lateral-torsional support at the middle. Here is the procedure for its definition:

1. Select the Design|Steel Design Data command to open the window where additional information about beam supports can be specified. See figure 3.15.

2. The Load application position is already correctly selected – it is unfavorable. The same is true for the shear web stiffeners – we don't have any. We only have to tell that the beam has an additional (ZZ – weak axis) and torsional support in the middle.

3. Press the ZZ (weak axis) button and a new window opens, figure 3.16. In this window we specify support positions and buckling koeficient between subsequent supports (usually 1.0). In this case, an additional support is needed.

4. Increase number of supports to 3 (Click up arrow once or enter number 3).

5. Enter 0.5 into the empty field that appeared in the Rel. pos row.
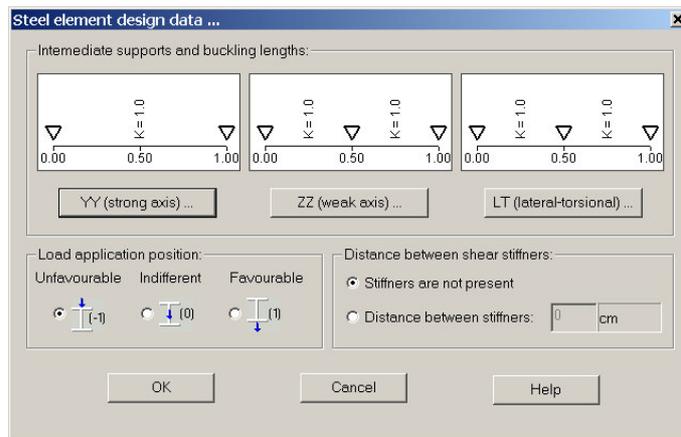
Figure 3.15: Steel design data window. Here we specify additional geometric information about individual steel element.
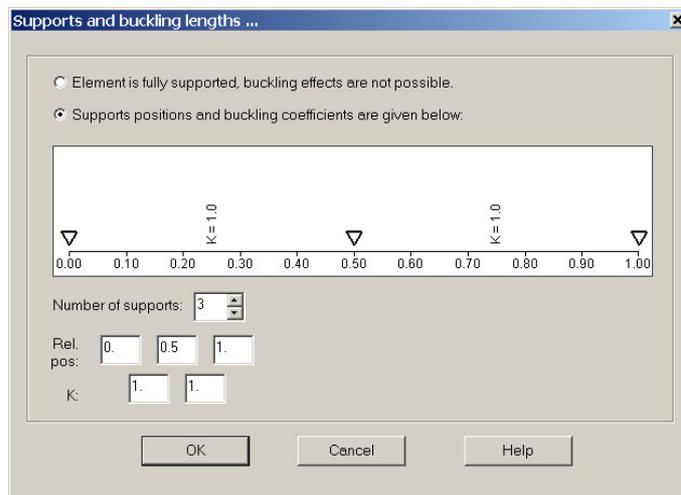
Figure 3.16: Support conditions, supports and buckling coefficients for the selected axis.

6. Hit OK to close the window.

7. We must repeat the procedure for the torsional-lateral support. Press the LT (lateral-torsional) button.

8. Increase number of supports to 3 and enter 0.5 into the new field.

9. Press OK to close the window.

10. Press OK once more to return to the steel design window.

Effective support width: The mathematical model of the structure is idealized in a wire frame model. This model neglects the actual depth of elements and replaces the cross-section shape with geometric properties. As a consequence, most elements length is slightly longer than the actual one, which leads to larger values for the internal forces. To compensate such effects and reduce internal forces at the element end, we may take into account the effect of the effective support width (or the depth of the

© 1999-2002 Ainet

connecting element). In usual case, the effective width is one half of the actual width, see the figure 3.17. However, there may be cases, where such width is different (zero, for example).

When the effective supporting width is given AMSES Frame2D cuts the internal forces on the supported areas, hence slightly smaller internal forces are used in the design.
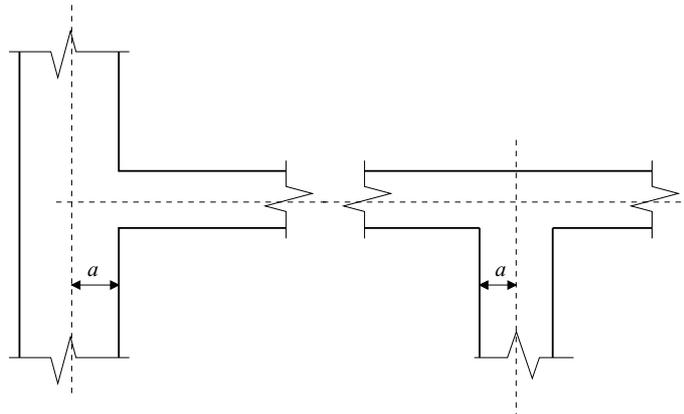
Figure 3.17: Effective supporting width for the beam-column connection case and intermediate support of the continues beam case.

If we are not sure about specifying the support widht, we can always assume zero width, which is a conservative assumption.

Element 3 gets 13 cm of support width on both ends.

1. Select the Design|Element support width command. The support width window is opened.

2. Enable support width effects.

3. Enter 13 cm into both fields.

4. Hit OK to close the window.

Ultimate limit state design: The beam geometry is completely set now. Let's check ultimate load cases for our beam.

1. Make sure that the U Ultimate tab is selected. Below the tab you see list of the load cases.

2. Select those load cases you want to include in the design. Press the Select [U] button to select all ultimate load cases in one step.

3. After a few moments results appear on the screen, figure 3.18. (If you do not see the results, press the ![icon] icon to recalculate internal forces and then the ![icon] icon to initiate the design.)

The top chart shows the class of each cross-section. Since several load cases are considered, the highes value is shown for particular point. Eurocode 3 declares four different classes. The values (bars) you see on the screen are all below 1, which classifies
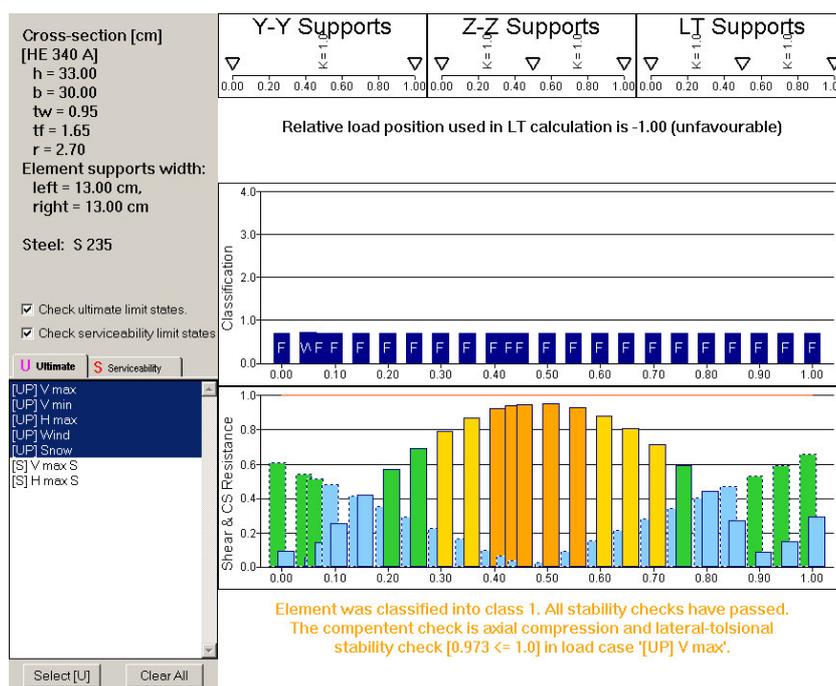
Figure 3.18: The situation of the ultimate design of the beam – element 3.

all cross-sections into class 1[4]. The letter F in the bars stands for the flange, which means that the flange has the highest slenderness in the cross-section[5].

The bottom chart displays exploration rate of cross-section resistance. Dotted bars refer to shear resistance while solid bars refer to complete resistance. Element passes the cross-section resistance test if all bars are smaller than 1.0. The color of bars differs according to the exploration rate:

- – light blue for interval [0.00,0.50),

- – green for interval [0.50,0.70),

- – yellow for interval [0.70,0.90),

- – orange for interval [0.90,1.00),

- – red for values > 1.000.

Obviously, bars should never be red. In our case we reach the *yellow* area.

Besides the individual cross-section whole element stability must be checked as well. Basically, three kind of checks are made for each element and they vary according to the distribution of internal forces and supports. This may result in a few dozens stability checks for each element. The result of the most relevant check is presented on the bottom of the screen in a textual form. The color of the text refers to the score.

---

[4] Values 0-1 refer to class 1, values 1-2 refer to class 2, values 2-3 refer to class 3 and values greater than 3 refer to class 4.

[5] Besides F we can meet letters W and O as well. The letter W stands for the web and indicates cross-sections where the web has the highest slenderness. The letter O may be met in the welded boxes. It stands for the outstand part of the flange.

53   © 1999-2002 Ainet

In our case all stability checks were passed and the worst cases scores 0.878, hence the text in yellow.

Serviceability limit state design: Serviceability checks are just a few clicks away. Click on the S Serviceability tab and the content of the screen is changed. Both charts are empty since no load case is selected.

Click the Select [S] button to select all serviceability load cases. After a moment both charts are shown and the screen should look like the figure 3.19.
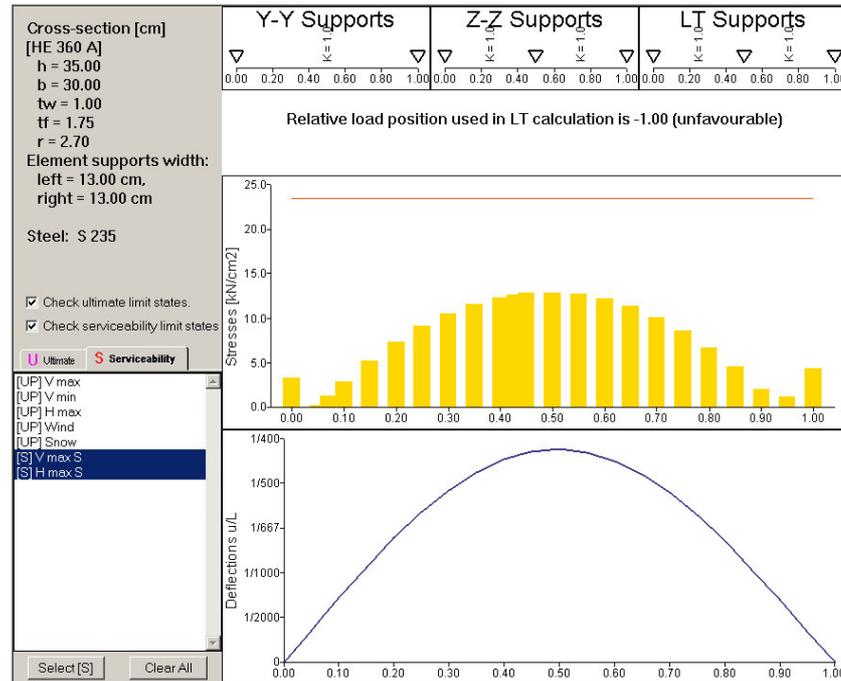
Figure 3.19: The situation of the serviceability limit state design of the beam – element 3.

The top chart shows envelope of the stresses for selected load cases. Cross-sections classified into class 1 and 2 may develop a plastic hinge in ultimate limit states, however in the serviceability limit states the stresses must not cross the f_y limit.

The bottom chart displays envelope of *local* displacements in the $\frac{w}{L}$ form. Please note, that global displacements, especially horizontal displacements, must be verified manually. Our displacement reaches ratio $\frac{w}{L} \approx \frac{1}{400}$.

A brief resume: The beam cross-section satisfies both limits states and has a slight reserve in capacity and displacements. You may consider replacing current HE 360 A cross-section with HE 340 A. The procedure is simple:

1. Click the ⤼ icon and select the HE 360 A cross-section.

2. Press the Edit button.

3. In the sections list browse for the HE 340 A cross-section.

4. Press OK twice to close both windows.

5. The cross-section has changed and the internal forces and design results are lost. Click the ▣ icon to calculate new internal forces.

6. Click the ▣ to redesign the beam.

The left column – element 1

The procedure to design this element is almost identical to the procedure used on the beam element. The only difference can be found in the supports and the load application position. This element has no intermediate supports and it has neutral load position. Additionally, we will not take effective support width into account.

1. Expand the Steel item and double-click the Element 1 line.

2. Select the Design|Steel Design Data command.

3. Move load application position into the indifferent state.

4. Press OK to close the window.

5. Press the Select [U] button to include all ultimate load cases into the design verification.

The results are shown on the screen. As you can see, the stability check has failed. This means that a stronger cross-section needs to be selected or an additional torsional support provided.

The right column – element 2

The design procedure for the right column is identical to the previous one.

3.2.7   Design of reinforced concrete beam

The foundation beam is designed in the very same way as it was presented in the first example – continues beam. Nevertheless, there if one important thing to keep in mind. The local coordinate system of the foundation beam points rightwards. This means that you must pay attention about the positive and negative side of the beam – the beam is turned upsidedown. E.g. the stirrups you define and see on the left are on the right in the reality.

Ultimate limit states

1. Expand the Reinforced concrete branch and double click the [04] Element.

2. We have already defined and selected the reinforcement material. If you forgot to do this, you can do it now. Click the ⤴ icon and select the RS 400 material.

3. Define stirrups ϕ 10 mm, e=12 cm for areas [0.0,0.2] and [0.8,1.0] and e=20 for area [0.2,0.8].

4. The negative reinforcement is minimal. Define four ϕ 12 bars that run over complete beam and specify 5 cm concrete cover.

5. Define four ϕ 28 bars over complete area for the positive reinforcement. Define 5 cm of concrete cover.

6. Press the Select [U] button to select ultimate load cases.

Results of the design are presented on the figure 3.20.
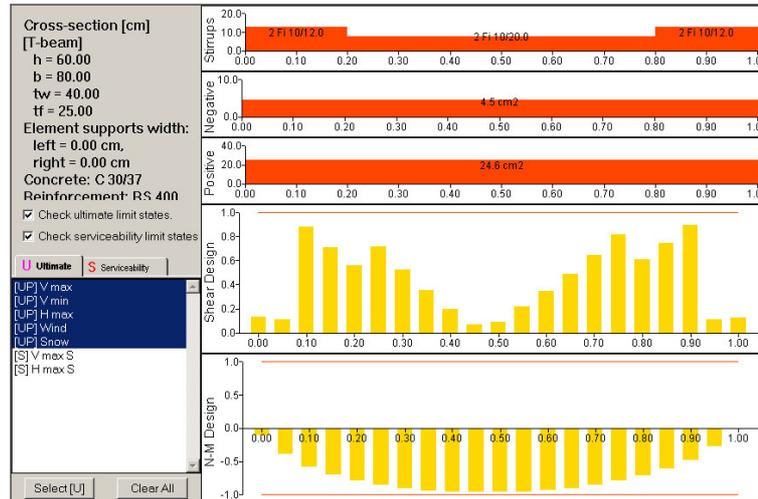
Figure 3.20: Results of the reinforced concrete design of the beam.

Serviceability limit states

The ultimate limit states are under control, but we must not forget to verify the serviceability limit states, too.

1. Press the S Serviceability tab.

2. Press the Select [S] button.

Opps, the crack widths are too large, more than 0.3 mm. Here we have two solutions. We may add more reinforcement or try to utilize more bars with smaller diameter (or both).

Try replacing 4 ϕ 28 bars with 7 ϕ 21 bars. The reinforcement area has actually decreased a little bit, however the cracks are now smaller than 0.3 mm and we may accept this solution (depends on the environment aggressiveness). If this is not enough, more reinforcement is needed.

3.2.8 Output settings

When we deal with more than one or two elements, we may consider to *optimize* the quantity of the output. It turns out that we do not need to print all tables and figures – some may be omitted.

Let's take a look how to fine tune the output. This can be done using the Tools|Output Settings command.

© 1999-2002 Ainet

- Select the General tab and make sure that the At least 3 pictures per page option is selected. This option commands AMSES Frame2D to condense the pictures as much as possible.

- Select the Geometry & Load tab. We can spare a page or two if we decide not to print tables of loads besides the load schemas. Uncheck the Tables option within the Print frame.

- Select the Results tab. Options stated here have the most influence on the output quantity. As far as internal forces are concerned, we do not need both: pictures and tables. Therefore, uncheck Displacements, Reactions, Internal forces and Envelopes to remove tables from the output. Figure 3.21 shows the Results page with modified settings.

- In the case of ultimate limit state load cases, we are mostly interested in internal forces and not in displacements, rotations and local displacements, which can be unchecked under the Ultimate column.

- In the case of serviceability limit cases, we are mostly interested in displacements and reactions, hence we uncheck everything but this option.

- Envelopes are used in rare cases. Mostly you will be interested in bending moment and thus uncheck everything but the My option.

- Additionally, you may completely remove some load cases from the output.

- Important: Design results will be printed only if the Print design results is checked.

In most cases we get significantly shorter output after these settings are applied.

Note: Before you actually print the results, it is wise to take a preview on the screen. If you want to change something, correct the settings and preview again. You may also consider to redirect the output into a postscript file or a pdf file and burn a CD instead of print out a pile of paper.

© 1999-2002 Ainet
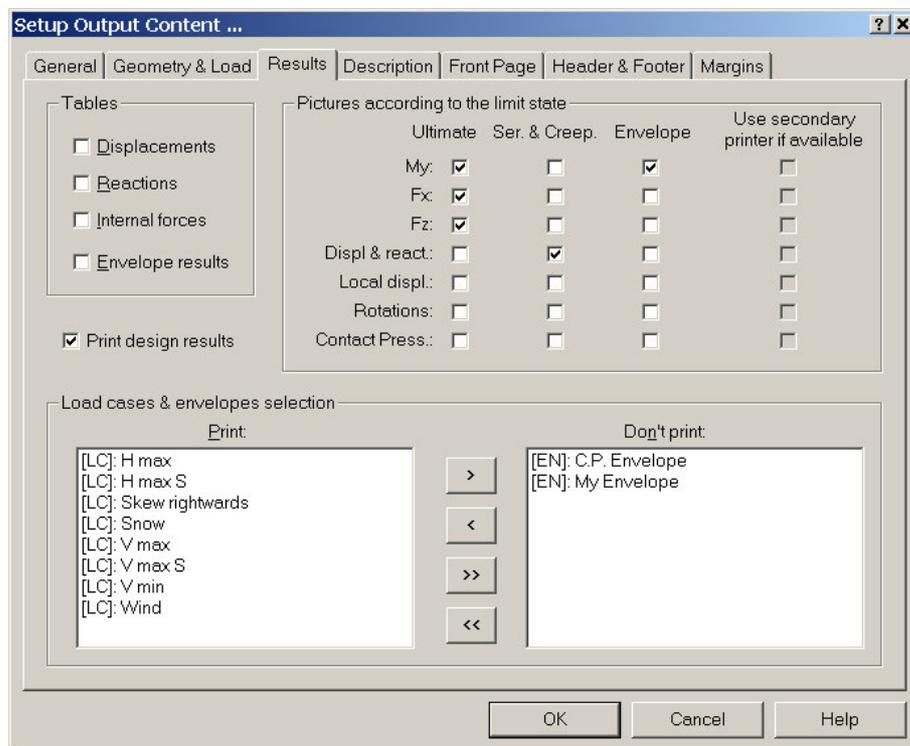
Figure 3.21: Recommended settings for the output.

CHAPTER 4

Basics

In this chapter we will reveal some of the AMSES Frame2D properties and details, which make the program powerful, elegant and useful.

4.1 Coordinate system

AMSES Frame2D uses a bit unusual coordinate system. The main axis are x and z, as shown on the figure 4.1. The program is using two coodinate systems: *global* and *local*.
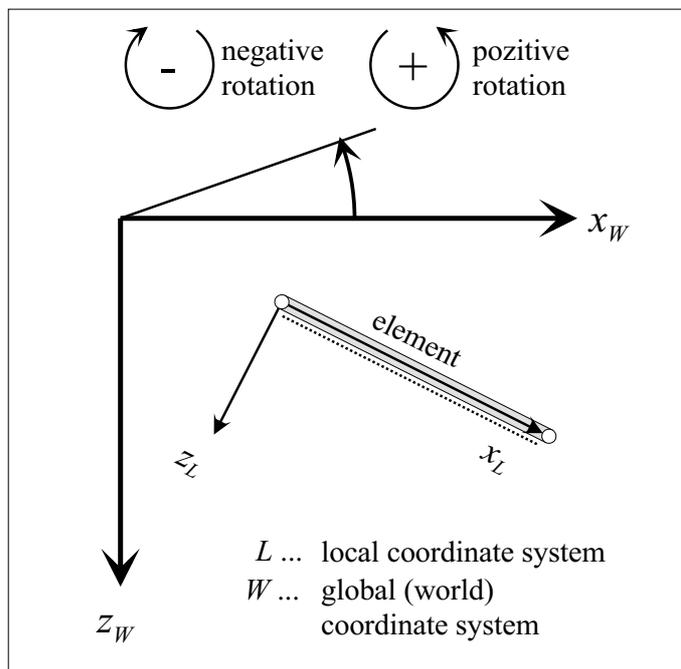
Figure 4.1: The illustration of global (world) and local coordinate system. The dotted line indicates the positive side of the element.

4.1.1 Global coordinate sistem

The global coordinate system is used mainly to describe the geometry of the structure and to declare directions for some of the loads. The x-axis is horizontal and points
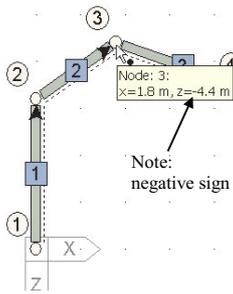
rightwards, while the z-axis is vertical and points *downwards*.

Such orientation has some advantages. First, all gravity loads are positive and second, the same coordinate system is used by Eurocodes. Unfortunately, there is one disadvantage – the structure *grows* into negative direction. E.g.: If the origin is located at foundations, then the top of the building has a negative z coordinate.

Rotation

The figure 4.1 also defines positive (counterclockwise) and negative (clockwise) rotations. This definition is valid for both coordinate systems.

4.1.2 Local coordinate sistem

The local coordinate system is related to an element and is defined by the element direction. The local x-axis points from the start node to the end node. AMSES Frame2D identifies the element direction with an arrow at the end node. The local z-axis is adjusted to the local x axis. The positive side of the element (positive local z coordinate) is marked with a dotted line that runs along the element, figure 4.2.

The positive side is important during the analysis where we must define the *positive* and *negative* reinforcement.
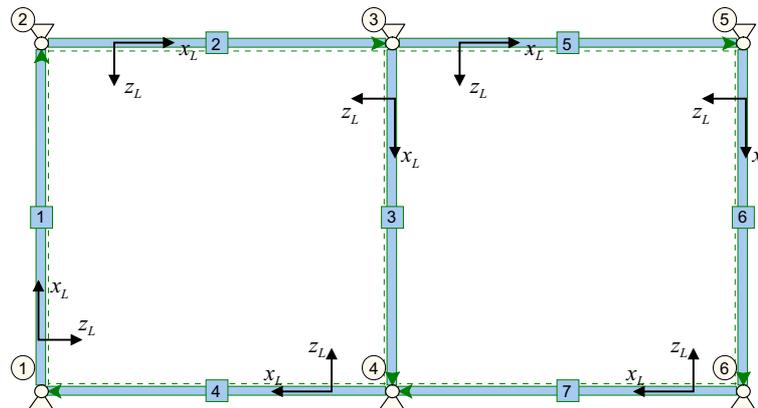
Figure 4.2: An example of local coordinate systems. Dotted lines represent positive side of the elements – positive side of the local z-axis.

4.2 Unit conversions

AMSES Frame2D has a unique feature, which can not be found in any other similar program. It can automatically convert between units of the same physical base. This enables us to enter data in units that we are most familiar with. These units are then automatically converted into the units needed by the analysis. When the analysis is finished, units are converted back to the desired units.

Example: Assume that we use *cm* for distance and *kN* for forces. If we want to enter some force per distance unit, then most computer programs require the *kN/cm* unit. However, this is an impractical unit since loads expressed in *kN/cm* are very small and we do not have the *right feeling* for them. To overcome this shortcoming, AMSES

Frame2D accepts any unit, *kN/m, kN/cm* or even *klbf/in*. All we have to do is to select the correct unit before we enter its intensity.

4.2.1 How to use the unit conversion

AMSES Frame2D uses the unit conversion feature wherever it is possible. In order to use this feature correctly, we have to follow one rule: *Select the unit first and enter its value later.* In most cases, default units suit our needs, but sometimes we need to enter a physical value in some other unit.

Select the unit first and enter its value later.

Let us illustrate the conversion feature using the following scenario: The default force unit is *kN* and default moment unit is *kNm*. Yet, we got the following concentrated force data: Fx = 5000 *N*, Fz = 10000 *N* and My = 30000 *kNcm*. The unit conversion feature enables us to enter such data directly.

1. Click on the kN label of the Fx field. A list of related units pops up; see figure 4.3.

2. Select the N Newton unit. Notice that Fx and Fz units are changed after the popup-menu selection.

3. Enter 5000 into the Fx field and 10000 into the Fz field.

4. Click on the kNm label of the My field and select the kNcm unit instead.

5. Enter 30000 into the My field.
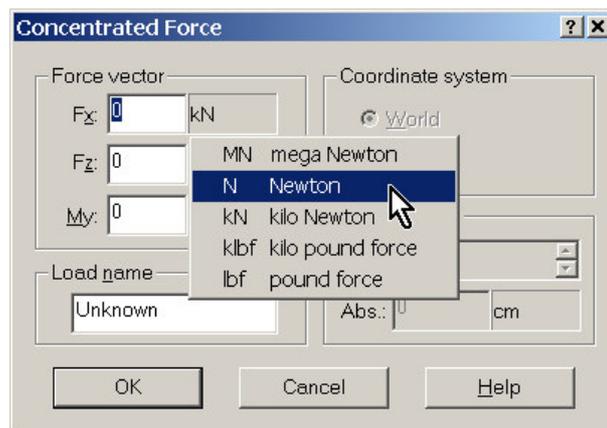
Figure 4.3: An example of unit conversion feature in action. Default *kN* unit was replaced with the *N Newton* unit.

In this example units of the Fx and Fz field were related. When we change the force unit of the Fx field, the unit of the Fz field is changed as well. This is a common practice within the program. On each window, most units that have the same physical base, are linked.

The change of unit in one field automatically reflects in a change of all linked fields.

4.3 Object properties

AMSES Frame2D uses a few major objects: nodes, elements, nodal loads, element loads and results. Each of these objects possesses important attributes, which can not be all presented on the screen. However, they can be presented in a special window called *property window*. The property window is opened if we are in the selection mode and if we double-click the object. Property windows usually consist of several pages in which we can modify the object attributes. There are a few simple rules about property windows that we should keep in mind:

- Double-click an object to open its property window.

- Right-click an object to pop-up a menu. The last menu item is the *Properties* item. If we select it, the property window appears.

- Recent changes of object properties are accepted and stored only after the Apply button is pressed. The button is located on the left side of the property window.

Press Apply to accept the changes.

- The button keeps the property window on the screen. If we click on some other object, the property window stays on the screen and properties of newly selected object appear in the property window.

- If property window is *pinned*, than several objects of the same type can be selected simultaneously. The modified attributes will be applied to all selected objects.

  Example: We selected elements 5, 7 and 9. Now we can change the material, cross-section properties, releases, etc. to all elements in one singe step. Note that there are a few attributes that are unique to each element and the program will not allow you to modify them when more than one element is selected.

- Some objects, e.g. nodal and element loads, do not allow multiple selection.

- When the property window is not pinned, a mouse click outside the window closes it.

4.4 Undo/Redo system

A very complex system, which allows us to *undo* or to *redo* individual commands, is build in AMSES Frame2D. The system keeps track of all issued commands that are related to the structure. Commands like zoom, workspace switching, etc. are ignored by the Undo/Redo system.

If we realize that the last command (or a few of last commands) was wrong, we can simply undo the command by pressing the icon or by typing the Alt+Backspace key combination. The program will take care for the necessary actions and will restore the previous situation. The figure 4.4 illustrates the system behaviour.

The icon is a shortcut for the Edit|Undo command.

We can also change our mind deciding that the undone command was a good one after all. All we have to do is to hit the icon or press the Ctrl + Y combination and the command will be redone.
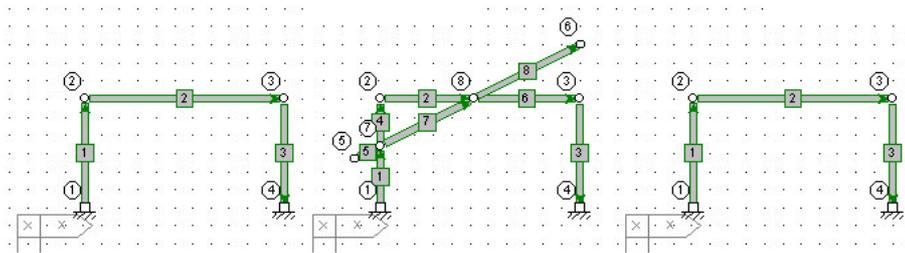
Figure 4.4: Initial state (left). A command spoils the situation (middle). The command is undone and initial situation is restored (right).

The level of Undo/Redo commands is unlimited. All important commands are recorded by the system.

Comment: Commands are undone/redone even if we switch to a different view or workspace. The switch is not recorded by the system. This means that some commands can be undone, but we will not be able to see the change on the screen.

4.5   View related commands

There are several commands, which are used to change the appearance of current view. The commands can be found on the View menu or on the top toolbar, figure 4.5.

Figure 4.5: View related toolbar.

4.5.1   Zoom commands

In many cases a structure that we are analysing is so big that individual elements appear too small, if the whole structure is shown on the screen. To overcome this problem, zoom commands were built in AMSES Frame2D. The commands can be found in the View|Zoom menu or in the view toolbar. The following zoom commands are available:

Zoom in makes the structure 50% larger.

Zoom out makes the structure 50% smaller.

Zoom window allows us to select the part of the structure to be enlarged.

Zoom previous returns zoom back to its previous state (a kind of zoom undo).

Zoom next reverses the last zoom previous command (a kind of zoom redo). Commands and are not limited to one level.

Zoom all shows complete structure on the screen.

Zoom extents fits the structure extents on the screen.

Each zoom command is a transparent command. It can be issued while some other command is active. E.g. you can zoom while dragging a new element.

© 1999-2002 Ainet

4.5.2   Other view commands

There are couple of useful view commands that can not be easily categorized and for this reason they are presented here.

The ⊛ button toggles the node labels.

Node labels: The View|Show|Node Index command is a toggle. First, it turns the node labels off the screen. If we repeat it, it will bring node labels back.

The ▣ button toggles the element labels.

Element labels: The View|Show|Element Index command is a toggle for the element labels.

The ⊕ icon expands the size of the nodes, labels and symbols.

The ⊕ icon shrinks the size of nodes, labels and symbols.

Label size: Icons ⊕ and ⊕ are used to change the size of symbols, nodes, element thickness, labels and size of text on the screen and output. They are very useful in cases where the size of a node is relatively big or small when compared to the whole structure. This usually happens when the structure is very small (a small truss) or the structure is very big or long (a long bridge). In the case of a small structure, nodes and symbols will be too big and we will reduce them by selecting the Geometry|Shrink Node command. In the case of a long structure, the nodes will be too small and we will expand them by selecting the Geometry|Expand Node command. We can use these commands several times in a sequence.

Commands *shrink* and *expand* annul each other. If we issue the first command 3 times and then the second command 3 times, we will get the same size of the nodes and symbols as they were at the beginning. These commands effects the output too. The rule of thumb is that if the picture on the screen looks nice, the printout will look nice as well.

Redraw a view: The View|Redraw command completely redraws the current view. This is useful if the drawing on the screen is corrupted for some reason.

4.6   Objects selection

There are only three visible objects: a node, an element and a load. In many cases, we need to select an object or several objects first and we issue some command on the selection afterwards.

Single object selection:

The ↖ icon switches into the selection mode.

1. Switch the program into the selection mode by pressing the ↖ icon (located on a toolbar between a view and tree pane) or by selecting the Edit|Select command.

2. Position mouse pointer over the object and click it. The object will change the color indicating that it is selected.

Hold down the SHIFT key to add new selection to the existing selection.

Selecting more than one object: The procedure for selection of several objects is similar - first two steps are the same. Now hold down the SHIFT key and click other objects. Object are added to the selection. If a selected object is clicked, it is removed from the selection (toggle behaviour).

Rectangle selection: A selection can be also made using a stretching rectangle. Here is the procedure.

1. Switch the program into the rectangle selection mode. Use the icon or issue the Edit|Select Window command.

   The icon switches into the rectangle selection mode.

2. Click to define the first point of the stretching rectangle.

3. Move the pointer to locate the second rectangle point. A stretching rectangle appears on the screen.

4. Click on the second point to define the rectangle. All objects located entirely inside the stretching rectangle are selected.

Comment: You can hold down the SHIFT key during the rectangle selection, too. All newly selected objects will be merged with any previously selected objects.

4.7   Objects deletion

It is simple to delete an object. Select an object (a node or element) and hit the Del key or select the Edit|Delete command.

If a node is deleted, all elements that refer to the node are deleted, too. The same is true for nodal and element load – if a node or element is deleted, their load will be deleted, too. We can undo the delete command.

Don't forget: we can undo the delete command

4.8   How to move a node?

We can move one node, several nodes, or even all structure at the same time. Here is the procedure we have to follow:

1. Be sure that we are in the selection mode. Select all nodes we want to move. (Do not forget to hold down the SHIFT key to select multiple nodes.)

2. Select the Edit|Move command or press the icon. This starts the move operation.

   The start the move operation

3. Click on a reference point.

4. Move the reference point to a new location. Notice that nodes and referred elements are moving along the mouse pointer. If you stop the movement, a tip window will display the difference between the current point and the first reference point.

5. Click on the second reference point to finish the move operation.

There is an alternative way to move a single node to a new coordinate. This way is usually used to specify a precise location of the node.

1. Right-click the node and select the Properties menu item.

© 1999-2002 Ainet

2. Select the Coordinates & Name tab.

3. Enter new coordinates of the node.

4. Press the Apply button to assign new coordinates to the node.

4.9   Some element specific commands

AMSES Frame2D uses two kinds of finite elements: linear and non-linear. Two kinds of finite elements are currently used in AMSES Frame2D: linear element and non-linear element. Both types are used in the same way and they look the same from the outside. However, they differ on the background theory. Generally, non-linear elements give more realistic results, especially in the case of sway frames. But linear elements are useful, too. They are typically used to model beam on *elastic subgrade* problems. Linear elements are also often used in educational process.

Since both elements are used in the same way, we do not distinguish between them in this section. Each element has following properties:

- An index that uniquely defines an element.

- An element name – any name can be used. It is used as a comment.

- Indices of start node and end node, which define the element position.

- Material index, which refers to material properties.

- Cross-section index, which refers to cross-section properties.

- Releases of individual degrees of freedom at either side of the element.

- Elastic ground properties that are used when the element is laid on the elastic subgrade (Winkler) halfspace. Note: This feature can be used with linear elements, only.

4.9.1   Orthogonal elements

When we work on a structure, which has orthogonal elements, we may turn on a feature which prevent us to draw inclined elements. The feature is called *ortho mode*.

We can toggle the ortho mode while we are dragging a new element. When the *ortho mode* is turned on, elements are drawn either in horizontal or in vertical direction, regardless to current mouse position. If we want do define inclined elements, we simply turn the *ortho mode* off by pressing the ⊡ icon in the middle toolbar or by selecting Geometry|Orto Mode. The ⊡ icon behaves like a toggle.

4.9.2   Element intersections

If we add a new element, which intersects one or more existing elements, nodes will be created at all intersections and the intersecting elements will be split at the intersection points. (The load that was originally put on the elements is split, too.) This default behavior can be altered if we hold down the SHIFT key while we are drawing a new element. In this case, the new element will be drawn over existing elements and nodes will not be created at intersections. The figure 4.6 illustrates both situations.
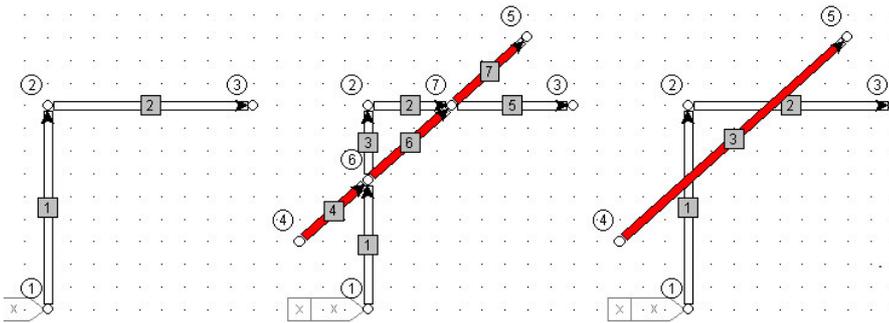
Figure 4.6: Existing elements (left). A new element crosses the existing elements. Hence, two new nodes and four new elements are created (middle). SHIFT was pressed while the new element was drawn. Hence, the new element bypasses exiting ones (right).

4.9.3 Releasing degrees of freedom

There are two general approaches for releasing individual degrees of freedom at the element edges:

- Condensation of element stiffness matrix.

- Applying a special kind of node called *linked node*.

We have used the linked node principle, since it is easier to implement it in the program code and because the linked nodes can successfully solve some additional problems. The disadvantage of this principle can be found in a fact that we can not release axial and shear forces on non-horizontal elements. In practice this does not present a problem, but it is a clear disadvantage from the educational point of view.

The program uses hidden linked nodes to release internal forces.

How do the linked nodes work?

The best way to answer this question is to look at an example. Let's suppose that we want to release bending moment at the end of a beam that is connected to a column. The situation is presented on the figure 4.7. In this case, we insert an additional (hidden) node (2), which has some of its degrees of freedom linked to the node (1). The node (2) is called the *linked node* and the node (1) is called the *master node* in this example.

Figure 4.7: Illustration of the linked node example.

Master and linked node usually (almost always) have the same coordinates. In order to speak about the linked node, the (hidden) linked node must have some degrees of

© 1999-2002 Ainet

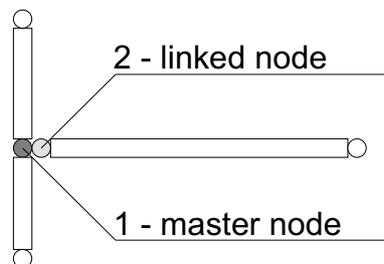freedom linked with the master node. If a degree of freedom is linked (bound), then any change of the degree of freedom on the master node will be identical in the linked node, too. In this case both nodes refer to the same equation in the stiffness matrix.

If a linked node has all its degrees of freedom linked with a master node, they occupy only three equations in the stiffness matrix. Actually, the linked node is redundant, since it has no degree of freedom of its own.

If one degree of freedom (e.g. rotation y) is released while others are linked to the master node, than both nodes occupy four equations. The master node occupies three equations and the linked node occupies one (rotation y).

If all degrees of freedom are released in the linked node, than there is no connection between master and linked node and the linked node behaves as an ordinary node. They occupy six equations. (This is more a theoretical than a practical case.)

Element displacements can be released only if the element lies parallel to world x-axis. Degrees of freedom of a node always refer to the world coordinate system. For this reason, we can release local u_x displacement and local u_z displacement only if the element lies horizontally. (The element local coordinate system is parallel to the world coordinate system.) If local u_x or u_z displacement ware released on an inclined element, then coordinate systems are not parallel and the results of calculation will be wrong. E.g. if we release u_x displacement (axial force) than you will still get some axial force in the element, although it should be zero.

Releasing rotations

Rotations can be released in any case. Rotations (bending moments M_y) can be released in any case, even for inclined elements. Why? y-axis of the local coordinate system is always parallel to the y-axis of world coordinate system and therefore no transformation is needed for rotations around y-axis. Fortunately, in vast majority of cases, we want to release rotations only.

The ⚙ icon toggles releases at the end of an element. In order to release an element, simply press the ⚙ icon. This puts us in the *toggle ϕ_y release mode*. Now all we have to do is to click on an element end, where we want to release the rotation.

4.9.4   Pinned node simulation

AMSES Frame2D does not support pinned nodes directly. If we must model such node, we need to simulate it. There are two options:
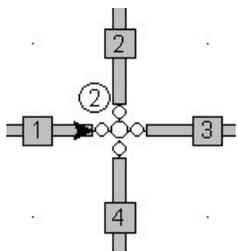
Option 1:   As it is shown on the figure, all elements that are connected to the node (2) use linked nodes which are linked to the master node (2). All linked nodes have their rotational degree of freedom released. In this case, rotation (referred as ϕ_y) in the node (2) is not used. This will be noticed by AMSES Frame2D during stiffness matrix set-up and a warning will be issued:

```
Solving ... Warning 1017: No equation was assigned for degree of
freedom (Ry) in node (2).
```

If we put moment load on node (2), we will get even more warnings. Namely, there is no way for the structure to resists such a load. Note that the structure will be calculated properly despite the warning – the moment load will be ignored.

Option 2: The second option is similar, with one difference. One of the elements (element 4) does not have linked node and it is directly connected to the node (2). All other elements are connected via linked nodes. This means that a rotation of node (2) reflects to the element (4) only. A bending moment in any element can't be transferred to any other element connected to node (2). Thus the second option is equivalent to the first. The node (2) acts like a true pinned node.

There is only one slight difference. In the second option, rotation of node (2) is used and has its own equation in the stiffness matrix. Additionally, if a bending moment is put on the node (2), the load will be accepted by the element four and no warnings will be produced.

We find the second option better, mainly because it does not raises false warnings.

© 1999-2002 Ainet
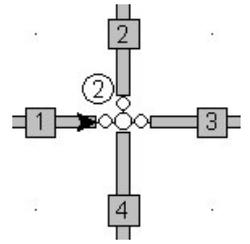
CHAPTER 5

Load

5.1 Basic definitions

When we talk about load we use a few terms, which are often used throughout the manual. These terms are *load*, *action* and *load case*. They are compatible with the terminology used by Eurocodes.

5.1.1 Load

Load is a generic name for everything that has a static or dynamic influence on a structure. Typical loads are: actual self-weight, actual dead load, actual traffic load, actual temperature load, actual prescribed displacements, reactions, earthquake force, explosion, etc.

5.1.2 Action

Action represents a group of actual loads, which have some common characteristic. Loads can be classified in many different ways. The most often used classification is presented below:

Action is a term for a group of loads.

- *Permanent actions (G):* Typical loads that fit into this group are self-weight of structure, installations, equipment, pre-stressing, etc.

- *Variable actions (Q):* Typical loads that fit into this group are traffic load, variable load, snow load, wind load, temperature load, etc.

- *Accidental actions (A):* Loads that fit into this group are rare, but have high intensity: explosions, fire, hit of a conveyance, etc.

- *Earthquake action (EA):* Dynamic load of an earthquake or its static equivalent.

Individual actions can be further divided into subgroups; this depends mostly on the problem we solve. In the most cases variable actions (Q) are subdivided into new actions: (S) snow actions, (W) wind actions, several positions of the live load (Q1), (Q2) ... It is up to the engineer how she/he will define the actions.

5.1.3 Load cases

Load cases combine different actions. A load case combines different actions into a realistic load combination. Each action in the combination gets its own safety factor. The values of safety factors are defined by building codes. A load case can be written in the following symbolic way:

*Load case = action 1 * factor 1 + action 2 * factor 2 + ... + action n * factor n*

AMSES Frame2D assigns a *limit state* type for each load case. Limit states are states beyond which the structure no longer satisfies the design performance requirements. Two basic kinds of limit states are used:

Ultimate – structural collapse or failure

Ultimate: Ultimate limit states are associated with structural collapse of with other form of structural failure. They generally concern the safety of the structure and its content and the safety of the people.

Load cases combine actions that may occur simultaneously into following situations:

- Ultimate P&T – (Persistent and Transient situations). Here are merged an dominant action and combination of other actions. This is the most common subtype of the ultimate limit state.
- Ult. Accidental – (Accidental situations). Permanent actions together with frequent values of the dominant variable action plus quasi-permanent values of other variable actions are combines with the design value of the accidental action.
- Ult. Seismic – (Seismic situation). Here we combine permanent actions together with quasi-permanent values of variable actions and the seismic action.

Reinforced concrete design applies different safety factors for different ultimate limit states. For this reason, it is advisable to choose correct ultimate limit state subtype.

Serviceability – service requirements faulure.

Serviceability: Serviceability limit states correspond to conditions beyond which specified service requirements for a structure or structural element are no longer met. Typically, the serviceability requirements concern the functioning of construction works or parts of them, the comfort of people, the appearance of the structure. Eurocode 1 gives a detailed explanation on limit states. As far as AMSES Frame2D is concerned, a more detailed definition is not needed.

For the serviceability load cases, we may additionally specify the creep and shrinkage coefficient ϕ_c that is used during the analysis and design (crack width).

Skewness is a special type of load case.

Skewness: load case type: simulates equivalent imperfections of the structures by skewing the structure for prescribed angle.

Creep: This kind of load case is obsolete. It will be removed in next program versions.

5.1.4 Safety factors

Each building code has its own approach for defining safety factors. However, they can be divided into two basic groups:

Load case	Type	ϕ_c	G	S	W
primary snow	Ultimate	–	1.50	1.50	–
primary wind	Ultimate	–	1.50	–	1.50
supplementary	Ultimate	–	1.33	1.33	1.33
displacements	Serviceability	–	1.00	1.00	1.00

Table 5.1: An example of global safety factors.

- Codes that use global safety factors.

- Codes that use partial safety factors.

Each approach has its own story, which is complicated and beyond the scope of this manual. Here is given a recipe, which tells how to use AMSES Frame2D for either group of building codes.

Global safety factors

When a global safety factor approach is used, it means that each action used in a load case is multiplied with the same (one global) factor. (Note that most modern building codes utilize partial safety factor approach.) Different global factors are usually used for different load combinations. We will show this on an example. Let us suppose that three actions have been defined:

- (G) permanent action,

- (S) snow action,

- (W) wind action

Now, suppose that building codes prescribe a global safety factor of 1.50 for the primary load case and 1.33 for the supplementary load case. Knowing this, we can set up the four load cases (we can set up any number of load cases):

1. Load case = 1.50 G + 1.50 S + 0.00 W

2. Load case = 1.50 G + 0.00 S + 1.50 W

3. Load case = 1.33 G + 1.33 S + 1.33 W

4. Load case = 1.00 G + 1.00 S + 1.00 W

The load cases above can be presented in a tabular form, table 5.1. Note that only one factor, but zero, is used for each load case.

Partial safety factors

Partial safety factors are used in most modern building codes, e.g. Eurocodes. The basic rule here is that each action used in a load case has its own factor. Since many different factors are used, one should have building codes at hand for a quick reference. *Refer to your building codes correct factors.*

Example: Let us assume that we have the same three actions as in the global safety factor case. Knowing this, we set up the four load cases (we can set up any number of load cases):

1. permanent + snow $= 1.35G + 1.50S + 0.90W$

2. permanent + wind $= 1.35G + 0.90S + 1.50W$

3. serv.: permanent + snow $= 1.00G + 1.00S + 0.50W$

4. serv.: permanent + wind $= 1.00G + 0.20S + 1.00W$

Comment: Safety factors used in both examples will not suit your situation and your building codes. Check out you building codes to find the right factors for your problem.

5.2   Equivalent imperfections

Logically, This section belongs into the geometry chapter, but we can benefit more if the equivalent structural imperfection is treated as a special kind of load case. This is why it is located here.

Geometry imperfection must be included in the analysis A good design practice requires that in the ultimate limit state, consideration shall be given to the effects of the imperfections in the geometry of the unloaded structure. Where significant, any possible unfavorable effect of such imperfections must be taken into account.

Different building codes estimate the structural imperfection, differently. However, almost all of them define the structural imperfection angle ϕ, for which the structure is inclined from the vertical (horizontal) axis, figure 5.1.

The usual measure for the angle ϕ is defined in a form $\phi = 1/X$, where X is some number, defined by the codes. Positive angle skews the structure in counter-clockwise direction and negative angle skews the structure in clockwise direction.

5.2.1   How to simulate the imperfection?

To handle the initial imperfection you may follow one of two basic approaches.

- Using the first approach, we define equivalent horizontal forces that simulate the inclination (skewness) of the structure. The horizontal forces are applied on nodes at each storey and they act on the ideally straight structure. By the ideal structure we mean a structure which is not inclined due to imperfections. This approach is assumed by most building codes. The drawback of this approach can be found in the fact that we must calculate the horizontal forces that are related to the vertical load on each storey.

© 1999-2002 Ainet

-φ   +φ

+   positive rotation

-   negative rotation
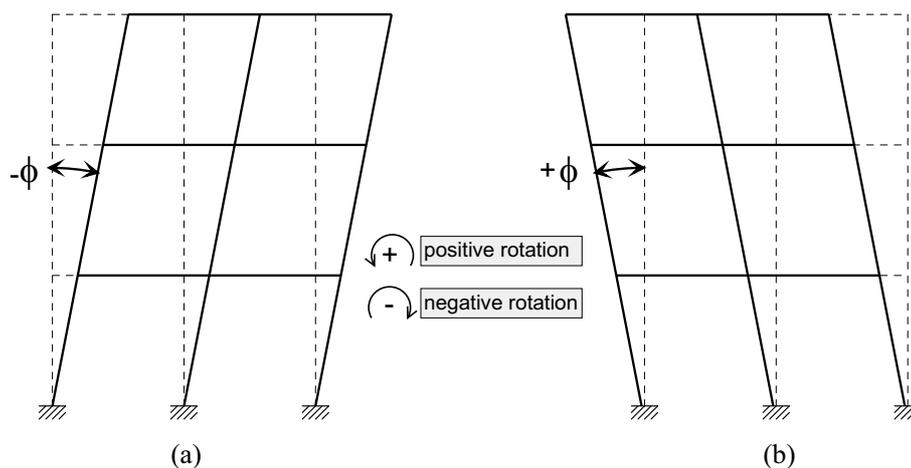
(a)   (b)

Figure 5.1: Illustration of equivalent imperfection. (a) Geometry is skewed rightwards – negative angle. (b) Geometry is skewed leftwards – positive angle.

- Usign the second approach we, specify the inclined (skewed) geometry of the structure. Since the structure is skewed, the effects of imperfections are in effect and there is no need to calculate equivalent forces. Unfortunately, this approach requires high precision on specifying node coordinates, which is tedious job.

We derived an approach, which is equivalent to the second one, but where an user is not required to specify precise coordinates of skewed structure. All she/he has to do is to specify the imperfection angle and AMSES Frame2D will do the rest: it will skew the structure, put on the load and calculate results - all behind the scenes. Furthermore, it allows you to specify different initial imperfections for different load cases. Namely, the structure may be non-symmetric and you can not know in advance, which skew direction will lead to the right effect.

In general, you can not know which skewed direction will give more unfavourable effect.

The skewness load case

We defined a special load case and named it Skewness. It can be defined in the Action/LoadCase Manager window. The skewness load case is a special load case that simply skews the structure for a given angle. All subsequent load cases are than evaluated on the skewed geometry. You can define any number of skewness load cases.

Example: Let's consider six load cases presented in the table 5.2. To simplify the example, only two action are used.

AMSES Frame2D will interpret the table 5.2 as follows:

Skew leftwards: Incline the structure for $\frac{1}{400}$ radians in the positive direction (counterclockwise). All load case factors are ignored and no other calculations are done.

Load case 1: Calculate the load case using specified factors and skewed geometry.

Load case 2: Calculate the load case using specified factors and skewed geometry. More load cases may follow here. The skewed geometry is used until another skewness load case is met.

Load case	Type	ϕ	S	W
Skew leftwards	Skewness	$\frac{1}{400}$	–	–
Load case 1	Ultimate		1.50	1.50
Load case 2	Serviceability		1.00	1.00
Skew rightwards	Skewness	$\frac{-1}{400}$	–	–
Load case 3	Ultimate		1.50	1.50
Load case 4	Serviceability		0.20	1.00

Table 5.2: An example os load case sequence when skewness load cases are used.

Skew rightwards: Reset previous imperfection angle. Afterwards, skew the structure for $\frac{-1}{400}$ radians in the negative direction (clockwise). All load case factors are ignored and no other calculations are done.

Load case 3: Calculate the load case using specified factors and skewed geometry.

Obtežni primer 4: Calculate the load case using specified factors and skewed geometry. Etc.

Comment: If you want to reset the imperfection angle, define Skewness load case and specify 1/0 for the angle. AMSES Frame2D will translate this into *zero* skew.

5.3   Concrete creep (time depended effects)

This section is mostly intended for concrete structures. If you are designing a steel structure you may want to skip it.

The concrete is one of the most often used building materials. But it is also one of the most problematic materials, as far as material models are concerned. It has visco-elasto-plastic behaviour, suffers from shrinkage effect and its characteristics are time, temperature and moisture depended. As a result, every author that had studied the behaviour of concrete proposed its own material model.

It is very difficult to write a computer program that will be able to take into account all concrete specific details. Most of the building codes are aware of this fact and they usually require that time dependent effects shall be taken into account where significant. Eurocodes also say, *"Creep and shrinkage normally need only be considered for the serviceability limit state, except where their influence in the ultimate limit state are likely to be significant."* Well, this does not help much, since no criteria for significance are given.

AMSES Frame2D uses concrete creep and shrinkage information during the analysis and design – crack width calculation. The procedure uses reduced elastic module of concrete $E_{c\varphi}$, which is defined as:

$$E_{c\varphi} = \frac{E_c}{1 + \varphi}$$

In this expression $\varphi \geq 0$ represents creep coefficient that reduces elastic module of the concrete. An evaluation of the φ can be found in relevant building codes. The typical values lie within the [0,3] interval.

         © 1999-2002 Ainet

Creep coefficient can be entered into the Fi creep/Skew column of the load case/action spreadsheet. This value will be used during the analysis and design. Creep coefficient is applied to the *serviceability* load cases, mostly. Namely, *ultimate* load cases simulate extreme, but short situations where creep does not have enough time to develop a significant effect.

5.4   Load and coordinate systems

AMSES Frame2D uses four different loads:

- concentrated load (forces and moment),

- distributed linear load (uniform load is just a special case of linear load),

- temperature difference load,

- load influenced by prescribed displacements.

5.4.1   Concentrated load

A concentrated load can be put on a node or on an element. When it is put on a node, its values F_x in F_z refer to the global coordinate system. However, when a load is put on an element, we can choose the coordinate system, figure 5.2.
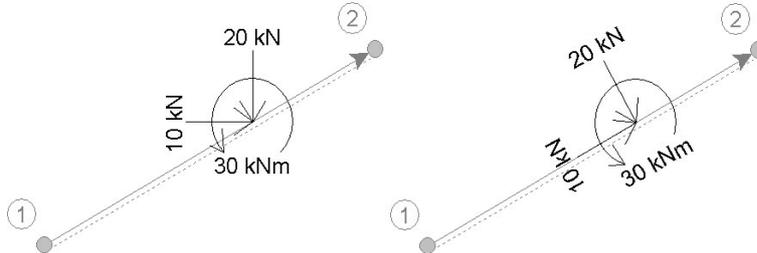

Figure 5.2: A concentrated load in the global coordinate system (left) and the same load in the element local coordinate system (right). The dotted line declares the positive direction of local z-axis.

5.4.2   Distributed load

A distributed load can be put on an element only. A special care need to be taken for correct selection of coordinate system for the load direction and load position. Internally, the program uses local coordinate system for both (direction and position) and the program will perform all necessary transformations, automatically.

As far as coordinate systems are concerned we meet three different basic types of distributed load:

- *Wind load.* Wind pressures always act perpendicular to the surface (element). Thus, their direction is local z-axis. When viscous wind forces are also considered (rarely) they act parallel to the surface – local x-axis. The load is also expressed in per structure surface area (element length) units.

© 1999-2002 Ainet

- *Snow load.* Gravity determines the direction of the snow load. Hence the global coordinate system is used for the direction. Intensity of the snow load is usually given in per ground surface area (baseline lenght). This means that the load position refers to the global coordinate system as well.

- *Self weight.* The load direction of the self weight is also governed by gravity. Hence the global coordinate system. However, the load intensity is expressed in per element length units and element's local coordinate system must be used to define the load position.

Now, let us take a closer look at each of distributed load coordinate system combination. We named the combinations by the most typical load. However, other kind of loads may use the same coordinate system combination.

Wind load – direction local, position local

The wind is actually a pressure that always acts perpendicular to the structure surface, hence it has *local* direction. This is true when viscose forces are considered, too (large roof areas and strong wind). The pressure refers to the local x-axis unit, which means that it lies in the *local* coordinate system, too. The figure 5.3 illustrates the situation.
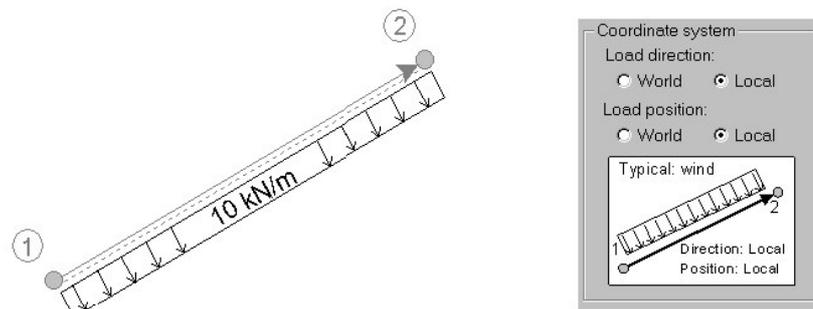

Figure 5.3: A load with local direction and local position. Wind load represents a typical case.

As far as finite element is concerned, it accepts the load without any internal transformation.

Snow load – direction global, position global

The snow load is a gravity load, hence the *global* coordinate system is used for the direction. The load refers to the x-axis unit of *global* coordinate system (basement line). The intensity of the load expressed in per global x-axis is always the same no matter how steep is a roof.[1] The figure 5.4 illustrates the situation.

Generally, only the vertical komponent s_z is given (its position is expressed in per global x-axis units). However, when the horizontal component s_x is given as well, it is expressed in per global z-axis units. And when the distributed moment m_s is specified as well, it always refers to the local x-axis.

---

[1] This claim is not 100% correct. Some building codes declares different values for the snow load that depend of the roof steepness. But the load is still expressed in per global x-axis units.
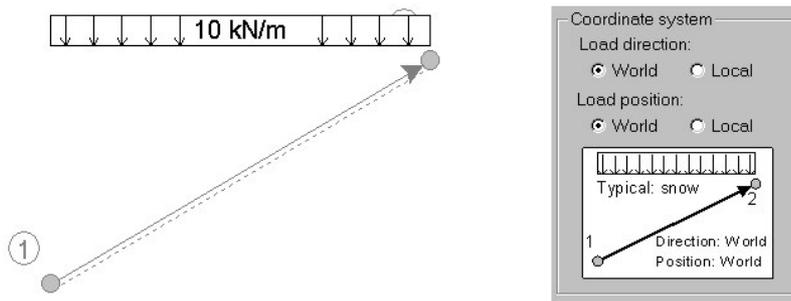
© 1999-2002 Ainet

Figure 5.4: A load with global direction and position. Snow load is a typical case.

Finite element can't accept this kind of load directly and the load needs to be transformed. Let's denote the load as $\vec{s} = \{s_x, s_z, m_s\}$. **AMSES Frame2D** transforms this load internally into the local coordinate system $\vec{s} \rightarrow \vec{p}$, where $\vec{p}$ denotes the load in local coordinate system. Let α denotes the angle between local x-axis and global x-axis. The transformation can be written using the expression 5.1. This expression is build into the program.

$$
\begin{aligned}
p_x &= s_x \sin^2 \alpha - s_z \sin \alpha \cos \alpha \\
p_z &= s_x \sin \alpha \cos \alpha + s_z \cos^2 \alpha \\
m_p &= m_s
\end{aligned}
\tag{5.1}
$$

Here we should bear in mind that the s_x component is used very rarely while the m_s almost never.

Self weight – direction global, position local

The self-weight is a gravity load, hence the *global* direction is used. The weight refers to the length of element, which means that it lies in the *local* coordinate system. The figure 5.5 illustrates the situation.

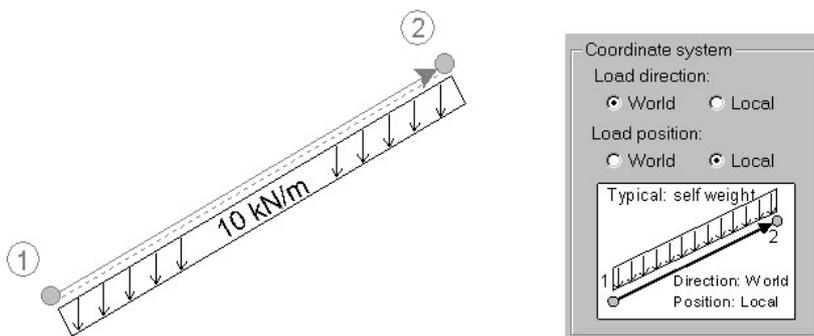Figure 5.5: A load with global direction and local position. Self weight (dead load) is a typical case.

Since the load direction is given in global coordinates, the transformation $\vec{g} = \{g_x, g_z, m_g\} \rightarrow \vec{p}$ is needed, expression 5.2.

$$
\begin{aligned}
p_x &= g_x \cos \alpha - g_z \sin \alpha \\
p_z &= g_x \sin \alpha + g_z \cos \alpha \\
m_p &= m_g
\end{aligned}
\tag{5.2}
$$

© 1999-2002 Ainet

5.4.3   Temperature difference load

We may assign to an element a temperature difference. A positive difference means that the element is warmer than some reference environment, while a negative difference means that the element is colder than the environment. Further more, we can specify different temperature difference on each side of the element and assume linear distribution of the temperature difference across the cross-section.

Let us take a brief example. A 10 m long, simple supported beam is made of hot-rolled IPE 360 steel cross-section. The beam is loaded with three temperature difference loads. Due to beam simple supports, internal forces do not appear, however the deflections do.

1. Negative side of the beam gets +30°C difference and the positive side gets +10°C difference. The figure 5.7 shows the window where the load was entered.

2. Negative side of the beam gets +10°C difference, while the positive side gets +30°C.

3. Both sides get +20°C difference.

Analysis results are presented on the figure 5.6. We used a non-linear element. The example file can be found in:

..\ProgramFiles\AMSES\Frame2D\Examples\Temperature.amp.
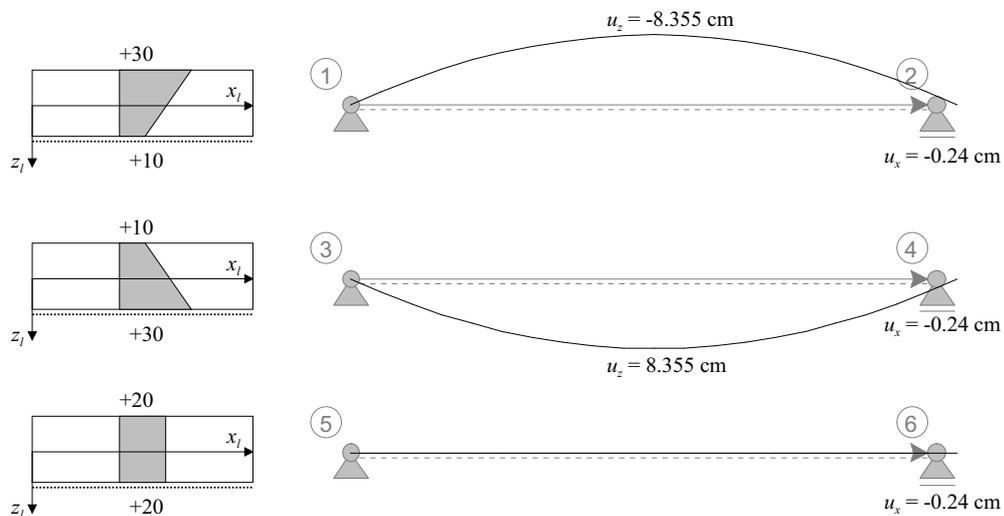

Figure 5.6: A beam loaded with three different temperature loads.

Temperature load window

In order to put a temperature difference load on an element, follow the procedure given below:

1. Select the Add|Temperature Load command or press the 🔥 button on the toolbar.

2. Click the element you want to load. The temperature load window appears.

ⓒ 1999-2002 Ainet

3. Enter the values of temperature difference into the appropriate fields. The fields are labelled as Neg. (-z) and Pos. (+z) which refers to the negative and positive element side, respectively.

4. Enter the load name (optional) and close the window with the OK button.

The load always applies to the whole element. To use the temperature load correctly we must know the direction of the element. The temperature load is in strong relation with the element material and cross-section properties.
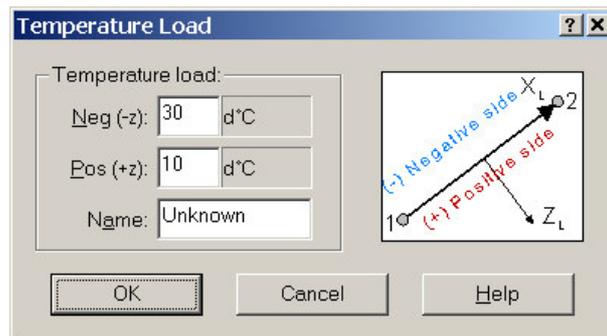

Figure 5.7: The temperature difference load window. We must enter the difference for the positive and negative side of element. Both sides can have the same differences, of course.

5.4.4   Prescribed nodal displacements

Prescribed displacements can be assigned to the supported nodes only:

1. Select the ⬆ icon or issue the Add|Prescribed Displacement command.

2. Click the node you want to prescribe an displacement.

3. Enter the size of the displacement. Note: you can only assign displacements (rotation) to supported degrees of freedom.

4. Press OK to close the window.

Comment: We can put concentrated force and prescribed displacement on the same node. Both loads will be correctly accepted and processed, however, their combination will not be drawn on the screen. It will be replaced with the Mix: U+F message.

AMSES Frame2D will not allow you to put the prescribed displacements on an unsupported node, but you can fool it and prescribe a displacement for unsupported degree of freedom.

Example: Select a node and support all degrees of freedoms. Prescribe displacements and remove the supports afterwards. In such a case, AMSES Frame2D ignores all prescribed displacements for unsupported directions and it issues several warnings.

© 1999-2002 Ainet

CHAPTER 6

General program settings – options

AMSES Frame2D lets you to fine tune some settings to meet your specific needs. Options are set in a special tabbed window. To open the window issue the Tools|Options command. Each tab represents a group of specific options.

6.1   Units

The units page is used to specify the units you want to work with. Any mixture of units can be used - there is no need to select compatible units. AMSES Frame2D will take all necessary steps for unit conversions. Selected units are stored on your disk as default units for all subsequent structures. Selected units will be used in the output, too.

Units are separated by their physical meaning.

The Load Metric Default button will set the units, that are most practical for the metric system (they are build into program) and the Load Imperial Default button will set units, that are most practical for the imperial system.

6.2   Precision

This page defines the numbers format. We can specify number of significant digits and the format type. We can ask the program to delete all redundant zeros, too. Two different precisions are used in the AMSES Frame2D:

- The working precision is used during the entering sessions in most dialog windows.

- The output precision is used for the printer output.

In both cases we define the same parameter: format, minimal numbers of significant digits and option to delete redundant zeroes.

The following formats can be used:

Smart: This format tries to present the output in a human friendly form. Frankly speaking, we have to polish this algorithm a little more.

Fixed: Number of decimal places is prescribed and scientific format is avoided if possible.

Scientific: All numbers are presented in the scientific (exponential) format.

The delete redundant zeros removes any meaningless zeroes form the number. This makes the output more compact and readable.

Filtration of small values

The structural analysis is numerically intensive process with many arithmetic operations. This usually introduces a small amount of error in the final results, especially when non-linear elements are used. The amount of such error is minimal, however it produces some undesired side effects in the output. Zeros are extra sensitive. In many cases values that should be zero are displayed as e.g. 1.432532E-14. The Threat values smaller than 1.E-X as zeros option allows us to overcome the undesired effect above. Values as 1.432532E-14 will be filtered out and replaced with a true zero on the output, which is what we wanted on the first place.

Comment: If we are working on a small structures and if we selected such units that produce results of very small magnitude, we have to either turn the filtration off or replace the threshold value with an appropriate.

6.3   Grid & Snap Units

The grid and snap units page is mostly related to the geometry workspace. The grid and snap settings help us to draw the structure element and nodes. On this page we can fine-tune the snap and grid settings.

Units type

Metric or Imperial units can be selected. Metric units are selected by default.

Grid & Snap

The major and minor steps are set here. We have to select the unit, size of the step in selected units and color of the step marker. The step size settings depend on the unit type. Following values can be used for metric and imperial unit system, respectively:

- Metric system: *cm* and *m* may be selected for the step unit. The step size can be 100, 50, 10, 5, 4, 2, 1, 1/2, 1/4, 1/5 and 1/10 of the selected step unit.

- Imperial system: *in* and *ft* may be selected for the step unit. The step size can be 12, 6, 4, 3, 2, 1, 1/2, 1/3, 1/4, 1/6 and 1/12 of selected unit.

Grid options

We can specify which step points are visible on the geometry window. Three options are given:

- Show major only displays major points only.

- Show major & minor displays major and minor points.

- Hide both does not display any points.

Important: The mouse input is always adjusted to the minor steps weather the step points are displayed on the screen or not.

Note: If the grid is too dense, the minor or even major grid points will not be shown on the screen. This usually happens to the minor points when working on large structures.

6.4   Personalize

This page is used to collect some information about you and your company.

Author name (your name): Enter your name or name of the person who will use AMSES Frame2D most often. Each time you start a new document (structure), the entered name will be used as the author name, automatically.

Company name: Enter name of the company you are working for. You can also leave the field empty if you like.

  This version does not allow specifying the company logos.

6.5   Symbol settings

The symbol settings options refer mostly to the symbols used to define elements of the structure.

General

The following options are considered as general:

- Font: The font typeface used for the labels and text, which is drawn on the screen. We recommend a typeface without serifs: ARIAL, TAHOMA, VERDANA, TREBUSHET, HELVETICA, . . .

- Selection color is the color used to indicate a selected object.

© 1999-2002 Ainet

Node

We can choose following graphical elements for a node:

- Line: the line color - color of the edge of a node circle.

- Label text: the text color used for a node label.

- Label area: color of the circle and label background.

Element

Similar attributes can be set for an element:

- Linear element: color of the linear element line (rectangle edge).

- Non-linear element: color of the non-linear element line (rectangle edge).

- Label text: the text color used for an element label.

- Label area: color of the label background.

Draw element as rectangle

In the geometry workspace, elements are drawn as rectangles by default. The color of the rectangle interior presents either element material or element cross-section. If you uncheck the Draw element as rectangle check box, the element will be drawn as a thin line. The second option may be useful on low resolution monitors. The figure 6.1 presents both possibilities.
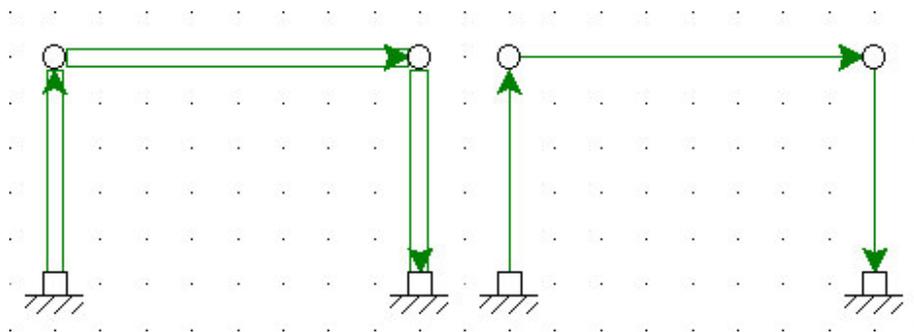

Figure 6.1: Elements drawn as rectangles (left) and elements drawn as thin lines (right).

6.6 Analysis

This page is used to specify some analysis parameters. Do not change the settings, if you are not sure about what are you doing.

Non-linear analysis settings

The following parameters governs the behavior of the non-linear analysis:

- Maximal number of iterations prevents an endless calculation. In most cases only a few iterations are needed to obtain the equilibrium. The default settings are 10 iterations. Rarely, when elements on elastic ground are used, this limit may be to severe and a larger value may be selected.

- Relative displacement precision is a parameter that serves as an analysis termination criteria for a load case. The load case analysis is terminated when relative difference of two subsequent solutions (nodal displacements) is smaller than displacement precision parameter. Value of 0.0001 may be used for the ordinary structures.

- Degree of interpolation polynomial. This is used for non-linear elements only. The default value is four. Specify greater value to get higher precision (and longer calculation times). We do not recommend values less than four, unless you want to experiment.

Element results

Number of internal points can be set here. The results are always calculated at the beginning and at the end of an element. However, this is not enough and results must be calculated in some internal points, too. The default setting is set to 9 internal points, which gives us 10 intervals.

Comment: The results are also calculated in some special points, which are related to the load that acts on the element. Such points are concentrated force application points and start and end points of distributed loads.

© 1999-2002 Ainet

CHAPTER 7

Element on elastic ground

Structural analysis always requires some idealizations of structure under consideration. The same is true for the foundations, too. Foundation idealizations in the sence of rigid, fixed, pinned and movable nodes are sometimes adequate, but there are also many cases, when they are not. This is especially true in situations, where upper part of the structure lies on beams, which are laid directly on the ground. Relationships between foundation ground and structure may be quite complex here. To solve many of problems of this type, this chapter introduces a special kind of finite element – element on elastic ground.

7.1 Some theory

Element on elastic ground is a special kind of finite element, that serves as a link between the upper structure and the foundation ground – subgrade . As its name suggests, it implies that the behaviour of the subgrade is elastic. In order to use such an element, we must know the *coefficient of subgrade reaction – k*. This coefficient tells us, how big is the contact pressure p between the beam element and the subgrade, if the element moves perpendicular towards the subgrade for one length unit (w). The mathematical model of this situation is presented on the figure 7.1.

The figure 7.1 shows two typical results. Figure (A) shows initial state. Figure (B) shows the case where the element stiffness is large enough to push complete element towards the subgrade. Figure (C) shows the case, where element stiffness is small comparing to the subgrade stiffness and only the middle part of the element moves towards the ground, while element ends are lifted off the ground.

Some parts of element on elastic ground may be lifted off the ground.

The relation between the displacement w and subgrade reaction is defined by the equation 7.1

$$k_t(x) = K \cdot w(x) \cdot b. \tag{7.1}$$

K - subgrade coefficient $[kN/m^3]$, w - displacement $[m]$, b - contact width between the beam and subgrade $[m]$ and $k_t(x)$ - subgrade reaction in the point x expressed in $[kN/m]$. Since tension between subgrade and the beam is not possible, we must slightly modify the previous equation:

$$k_t(x) = \begin{cases} K \cdot w(x) \cdot b & \ldots \quad w(x) \geq 0 \quad \text{– towards the subgrade} \\ 0 & \ldots \quad w(x) < 0 \quad \text{– lift of the subgrade} \end{cases} \tag{7.2}$$
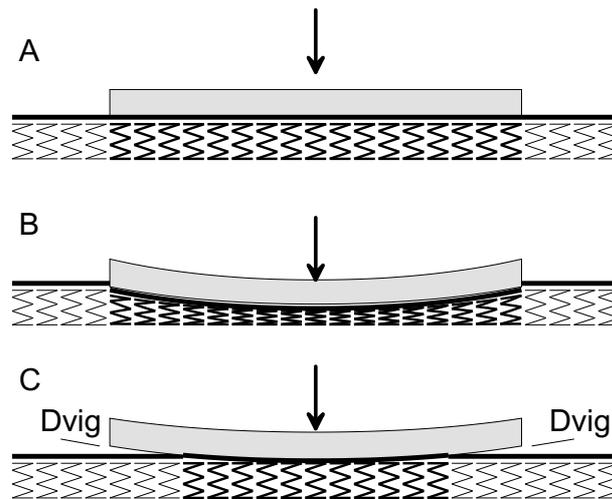
Figure 7.1: Illustration of the element laid on elastic subgrade.

As we can see, the pressure has bilinear behaviour; it is linear if the beam moves toward ground and constant (zero) if beam is lifting off the ground.

If we want to handle both cases (B) and (C) from the figure 7.1, an element must be able to identify all x points where $w(x) = 0$ and exclude lifted parts of the element. In general, this is a tricky job, especially for non-linear elements. Namely, it is difficult to identify all zeroes of the $w(x)$, since $w(x)$ may be complicated function, which consists of several high degree polynomials. Therefore, we decided to use linear-elastic finite element that has analytic solution for $w(x)$, which is a polynomial of third degree. For this polynomial, we can easily identify all x points for which $w(x) = 0$.

7.1.1   Displacements error

AMSES Frame2D internally splits each element on elastic ground into three finite elements to reduce the displacements error.

Since the element on elastic ground is based on the linear-elastic element, some error is present in the displacement $w(x)$. According to the Galerkin method (finite element method) the error can be minimized if more finite elements are used. AMSES Frame2D automatically splits every element on elastic ground into tree hidden finite elements to minimize such error. The splitting is not visible from the outside – it is done internally. All you can notice is that more degrees of freedom are used by calculation, as one would expect.

7.1.2   The solution of the problem is non-linear

Although the element on elastic ground is linear, the solution of the problem may not be linear. If the case (C) – lift off the ground happens, several iterations are needed to calculate contact areas properly. In the case (B), only one iteration is needed to obtain the solution.

Convergence rate is linear - quite a few iterations are needed if element lift off occurs.

In the case (C), the convergence rate is linear. This means that quite a few iterations may be needed to find equilibrium. You should not confuse this with the convergence rate of elastic-non-linear elements, which have quadratic convergence.

7.1.3   Nodal supports

Element on elastic ground resists in a direction which is perpendicular to the element axis. It has no resistance in direction, which is parallel to the element axis. This means that in most cases additional nodal supports are needed. A case like this is given in the figure 7.2. In this case, the structure is unstable if nodal support on the right is omitted – it can not resist against horizontal forces.

Figure 7.2: An example of structure that requires nodal support.

On the other hand, we can construct structures that need not to be supported with classical node supports. One example is given on figure 7.3. AMSES Frame2D will handle structures like this, if the coefficient of subgrade reaction is properly set. In the case shown on the figure, the sum of vertical load must be positive. If *lift-off* load dominates over the *gravity* load, the structure becomes unstable.
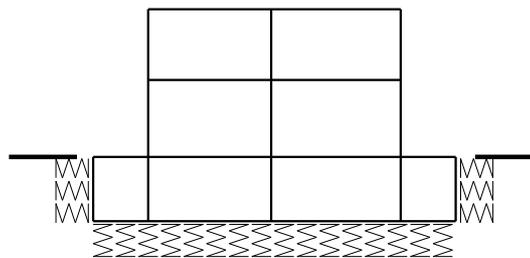
Figure 7.3: An example of structure without nodal supports.

As the last example, we show a pile, which has ground support on both sides and vertical node support at the bottom. In this case lifting off the ground is not possible and therefore the problem is linear. See the figure 7.4.
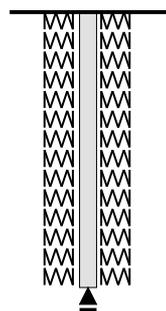
Figure 7.4: An example of a pile supported on both sides of the element.

7.1.4   Approximate values for the coefficient of subgrade reaction

The best way to get coefficient of subgrade reaction is to ask one of the qualified personnel (geo-mechanics engineer, for example) to provide it. She/he will take a look at the building site, take some ground samples or even drilling will be necessary, to get

Rigidness	Material type	K [kN/cm^3]
Very compressible	sand, clay	0.001 - 0.005
Semi-compressible	sand, clay	0.005 - 0.05
Compressible	sand, gravel, compact clay	0.05 - 0.1
Semi-rigid	compact sand, gravel, incompressible clay	0.1 - 0.2
Rigid	aggregated sand and gravel	0.1 - 1
Rock	monolith (solid) rock	1 - 5

Table 7.1: Approximate values for the coefficient of subgrade reaction.

a good picture. However, if you know that ground is compact and homogenous, you may consider values from the following table.

Please note that these values are approximate and more precise values are usually required. Several articles state that coefficient of subgrade reaction is not solely a ground characteristics and depends on the beam breadth, too (Terzaghi, 1959).

© 1999-2002 Ainet

CHAPTER 8

Linear and non-linear elements

8.1 General

AMSES Frame2D uses two types of elements: linear and non-linear. You work with both elements in the same way – they differ in their colors, only. Although they are almost the same on the outside, they are significantly different on the inside. The non-linear element is more complex and precise. The difference between elements can be noticed in structures, which are slender and have significant displacements – e.g. sway frames. Since non-linear analysis fulfils equilibrium equations in deformed state, we get more precise results than results obtained by linear analysis.

The non-linear element used in AMSES Frame2D is very reliable and heavily tested. We believe that there is no reason to use linear elements ever again (except for the beam on elastic ground) and this is true for any kind of structure you analyse.

Non-linear elements give better results than linear elements.

The program allows you to mix linear and non-linear elements in the same structure. The non-linear analysis will be used, if at least one non-linear element is used in the structure. Although element mixing is possible, it is not recommended, since results of the analysis are not so precise as they would be, if only non-linear elements were used.

Comment: The second order theory has limitations, but kinematically exact element is without any limitations. This means that large strain / large displacements discussion is irrelevant here and you may use AMSES Frame2D's element without any hesitations.

8.2 Three kinds of non-linearities

There is a little confusion about terms used in linearity and non-linearity discussions. To make things a little more clear, as far as this document is concerned, we will make a few definitions.

- *Kinematic (geometric) non-linearity* is related to the kinematical equation used to formulate the finite element.

- *Material non-linearity* is related to the stress-strain formulation (material model).

93

- *Other non-linearities:* Non-linear boundary conditions are the most typical representatives here. We have already met one – beam on elastic ground. The bilinear behaviour of the subgrade is nothing but a non-linear boundary condition.

8.2.1   Kinematic (geometric) non-linearity

This non-linearity is related to the kinematic equations of the planar element. The kinematic equations, derived by Reissner [4], are given below:

$$1 + u' - (1 + \varepsilon)\cos\varphi \;=\; 0 \qquad\qquad (8.1)$$
$$w' + (1 + \varepsilon)\sin\varphi \;=\; 0. \qquad\qquad (8.2)$$

where $u(x)$ is displacement in the x direction, $w(x)$ is displacement in the z direction, $\varepsilon(x)$ is extensional strain and $\varphi(x)$ is rotation of the centroid axis at x.

AMSES Frame2D uses the exact formulation (8.1 and 8.2) in its non-linear element.

First order theory

First order theory is obtained by limiting itself to small displacements, rotations and deformations, where we get: $cos\varphi \simeq 1$ and $sin\varphi \simeq \varphi$. Additionally, term $\varepsilon \cdot \varphi$ is very small and can be neglected. The simplified kinematic equations now become (8.3) and (8.4):

$$u' - \varepsilon \;=\; 0 \qquad\qquad (8.3)$$
$$w' + \varphi \;=\; 0 \qquad\qquad (8.4)$$

We see that equations are independent and may be treated as simple differential equations.

Second order theory

People were not happy with the first order theory, especially with the simplification of term $cos\varphi \simeq 1$. A better approximation is: $\cos\varphi \simeq 1 - \varphi^2/2$, which after neglecting small terms ($\varepsilon \cdot \varphi$ and $\varepsilon \cdot \frac{\varphi^2}{2}$) yields the equations (8.5) and (8.6).

$$u' - \varepsilon - 1 + \varphi^2/2 \;=\; 0 \qquad\qquad (8.5)$$
$$w' + \varphi \;=\; 0 \qquad\qquad (8.6)$$

This theory is limited to *moderate* rotations. The equations are depended – in the case of pure bending we also have horizontal displacements.

Co-rotational theory

The second order theory is better than the first order theory, but large displacements are still problematic. To overcome this problem a *co-rotational* theory was derived. This theory takes global rotation ($\Psi = $ const) of the finite element into account – equations 8.7) and (8.8).

$$u' - \varepsilon - 1 + \varphi^2/2 + \cos\Psi \;=\; 0 \qquad\qquad (8.7)$$
$$w' + \varphi + \sin\Psi \;=\; 0 \qquad\qquad (8.8)$$

© 1999-2002 Ainet

Exact kinematical equations

The co-rotational theory is very good, but not so good as the exact theory, which is used in AMSES Frame2D. There are no simplifications of initial Reissner equations here (equations 8.1 and 8.2) and as far as the kinematics is concerned, the element does not have any limitations.

8.3   Some illustrative examples

In this section we will take a look at three simple, but illustrative examples:

- The first example is a vertical cantilever, subject to vertical and horizontal load at the free end. This sample clearly demonstrates non-linear effects on sway structures.

- The second example shows high accuracy of the non-linear element used in AMSES Frame2D.

- The third example demonstrates that non-linear elements require meaningful cross-section and material data, but linear elements do not. So, if you play with non-linear elements and you get odd results, make sure, that you have specified correct cross-section dimensions and material.

8.3.1   Vertical cantilever – sway structure

We have two eight meters long cantilevers, made of steel S235 and IPE 240 hot-rolled cross-section. The free ends of cantilevers were loaded with axial forces of 50kN and lateral forces of 10kN. The left cantilever was modeled using linear element and the right cantilever was modeled using non-linear element. The figure 8.1 shows the cantilevers and their load.
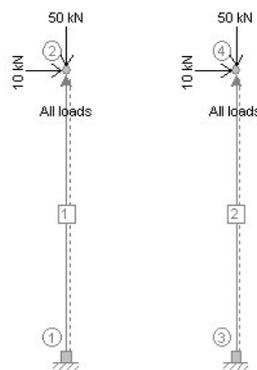

Figure 8.1: Cantilevers and their load.

The results of analysis are presented on the figure 8.2. Bending moments were calculated at safety factor of 1.50. The results for linear element can be easily verified: 10 kN * 8 m * 1.50 (factor) = 120 kNm, which is the same result as shown on the left cantilever. Surprisingly, we got significantly larger results for the right cantilever, which

Note that the program correctly analyses both cantilevers although they have no common nodes or elements.
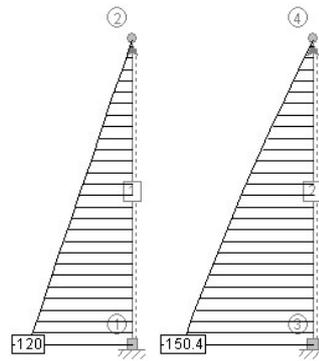
© 1999-2002 Ainet

Figure 8.2: Bending moments shown for both cantilevers. Linear and non-linear
analysis were used for left and right cantilever, respectively.

is made of non-linear element. Since non-linear analysis takes displacements into account, axial load contributes to bending moment, too. This makes non-linear result 25.6% larger.

The figure of displacements presents similar situation. Free edge of linear element moves 31.34 cm towards the X-axis direction, while non-linear element moves 40.85 cm. Displacement of the right cantilever is larger due to larger bending moments inside the element. The non-linear displacement exceeds the linear displacement for 30.3%.
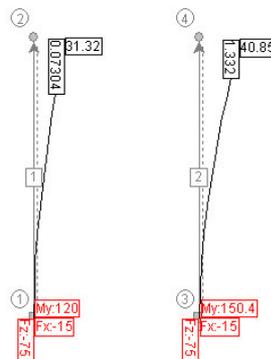

Figure 8.3: Displacements shown for both cantilevers. Linear and non-linear analysis were used for left and right cantilever, respectively.

8.3.2   High accuracy – horizontal cantilever and bending moment

The aim of this example is to show high precision of AMSES Frame2D non-linear finite element. The figure shows 8.4 the disposition of the element and moment load. The moments are calculated as $M_1 = \frac{\pi EI}{L}$ and $M_1 = \frac{2\pi EI}{L}$.

The analytical solution of the first moment results in a half-circle while the second moment results in a perfect circle.

The table reveals 8.1 results we got using only ONE AMSES Frame2D non-linear element compared with results that were obtained using other theories (first order, second order and co-rotational). Numbers in parentheses represents number of finite elements used to model the problem.
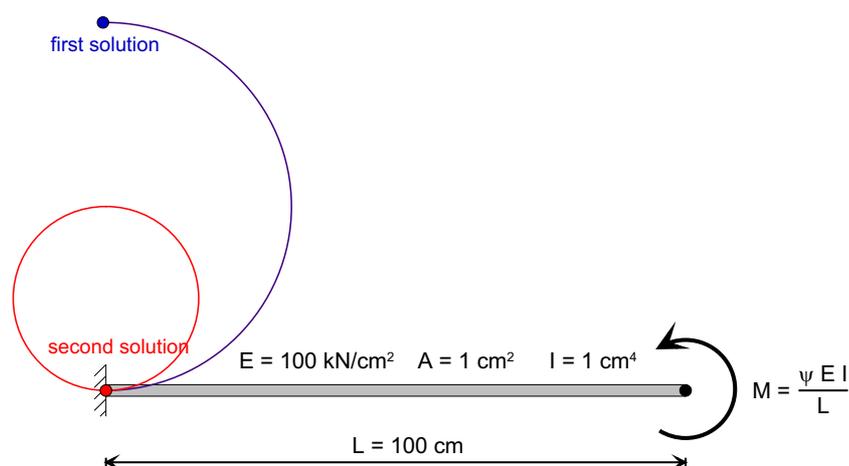
© 1999-2002 Ainet

Figure 8.4: The problem disposition, load and illustration of both analytical solutions.

Element type (theory) kinem. eq. type	$M_1 = \frac{\pi EI}{L}$			$M_2 = \frac{2\pi EI}{L}$		
	x	z	E [%]	x	z	E [%]
Analytical	-100.00	-63.662	—	-100.00	0.00	—
First order	0.00	-157.080	146.741	0.00	-314.159	—
Second order	-164.49	-157.080	146.741	-657.97	-314.159	—
Co-rotational (2)	-100.00	-64.119	0.718	—	—	—
Co-rotational (10)	-100.00	-63.663	0.002	—	—	—
AMSES Frame2D (1)	-100.00	-63.662	0.000	-100.00	0.000	0.000

Table 8.1: Results of the test using different kinematic theories.

The *first order* theory fails completely. We do not get any horizontal displacement (but we should) and we get an enormous vertical displacement (but we shouldn't).

The *second order* theory is just slightly better. Both displacements are wrong completely, but at least we got some horizontal displacement. True second order theory is of no use here.

The *co-rotational theory* managed to solve the first problem with sufficient accuracy, but we had to apply more than one finite element. The program failed to solve the second problem – the rigid body rotations were too large.

AMSES Frame2D non-linear element based on *exact theory* solved both problems without any problems and with very high accuracy.

8.3.3   Influence of dummy values for A, I or E

Non-linear elements are very sensitive to the geometrical properties E, A and I, but linear elements are not. To illustrate this fact, the following example was prepared, figure 8.5.

Both elements are fully supported and have exactly the same attributes. The length of both elements is 200 cm, elastic modulus $E = 1000 \text{ kN/cm}^2$ and geometric properties are chosen as $A = 1 \text{ cm}^2$ and $I_y = 1 \text{ cm}^4$. Both elements are loaded with linear load of
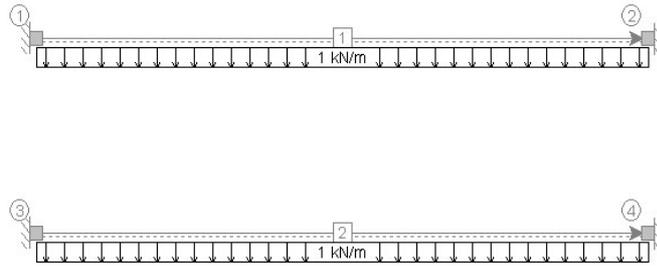
Figure 8.5: The disposition – top element is linear and the bottom element is non-linear.

1 kN/m. As you see, A and I_y are non-realistic. (If we choose E=1 kN/cm^2 non-linear element will produce useless results.)

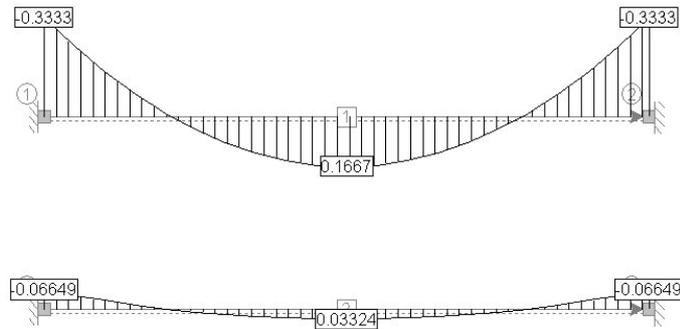In addition, here are the results. Bending moments are shown on the figure 8.6.

Figure 8.6: Bending moments M_y.

As we can see, the linear element calculated results that we are used to – but they are completely wrong. Namely, I_y is so small that element behaves as a rope rather than as a beam.
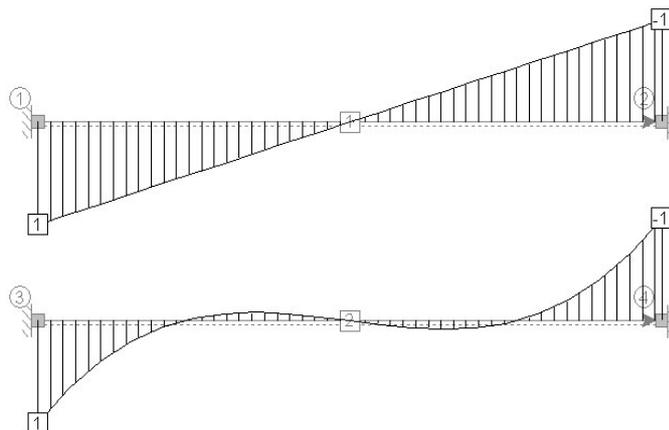

Figure 8.7: Shear forces F_z.

Shear forces are quite interesting, figure 8.7. We should not forget that axial and shear forces for the non-linear element are shown on undeformed geometry, but are calculated on deformed geometry and their orientation is valid for deformed geometry.
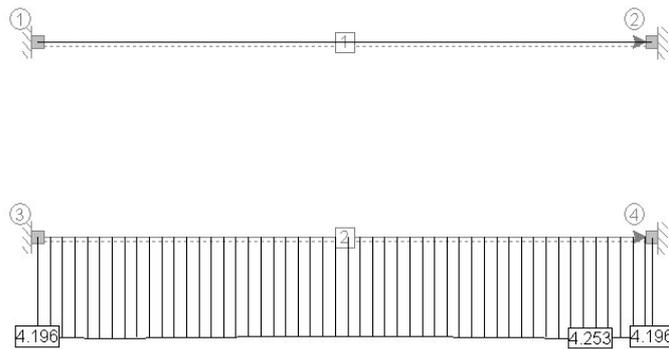
Figure 8.8: Axial forces F_x.

Axial forces – the effect of the rope is clearly visible on the figure 8.8. The linear element has no axial forces, while the non-linear has quite significant axial forces.
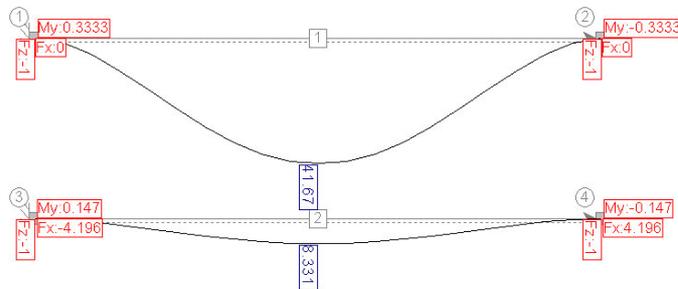

Figure 8.9: Displacements u_z.

The figure 8.9 shows the displacements. They are totally unrealistic in the linear case, but very precise and logical in non-linear case.

© 1999-2002 Ainet

Bibliography

[1] CEN. *Eurocode 2: Design of concrete structures – Part 1: General rules and rules for buildings.* European Committittee for Standardization, 1991.

[2] CEN. *Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings.* European Committittee for Standardization, 1992.

[3] CEN. *Eurocode 1: Basis of Design and Actions on Structures – Part 1: Basis of Design.* European Committittee for Standardization, 1993.

[4] E. Reissner. On one-dimensional finite strain beam theory: the plain problem. *J. Appl. Math. Phys. (ZAMP) 23*, pages 795–804, 1972.