

# **STRAD**

## **Quick Start Guide**

1. Installation – Launching
2. Working with STRAD



# Preface

This Quick Start Guide provides a fast and friendly introduction on STRAD main features and functionalities. All the features and functions of the program are presented and explained in detail within the complete User's Guide, along with informative examples.

STRAD is a powerful software tool for Structural Analysis and Design of 3D Concrete Frames. It has a friendly 3D CAD environment based on IntelliCAD (an AutoCAD-like CAD environment), a powerful finite element analysis engine performing both static and modal (eigenvalue) analysis and a reliable design module for concrete members, highly customizable to accept the parameters of any seismic code.

This guide aims at a short introduction on STRAD, showing its concept and basic operation principles. More specifically, it shows the user how to proceed with the installation first and then how to get familiar through a step-by-step example. A short introduction to terminology issues is also provided before the example.

Despite the simple example and the small part of the features and real capabilities of the program in fact, the user can get a good idea about STRAD environment, which is anyway the main purpose of this quick guide. Besides, all the features and instructions of STRAD are described in detail within the complete user's guide.



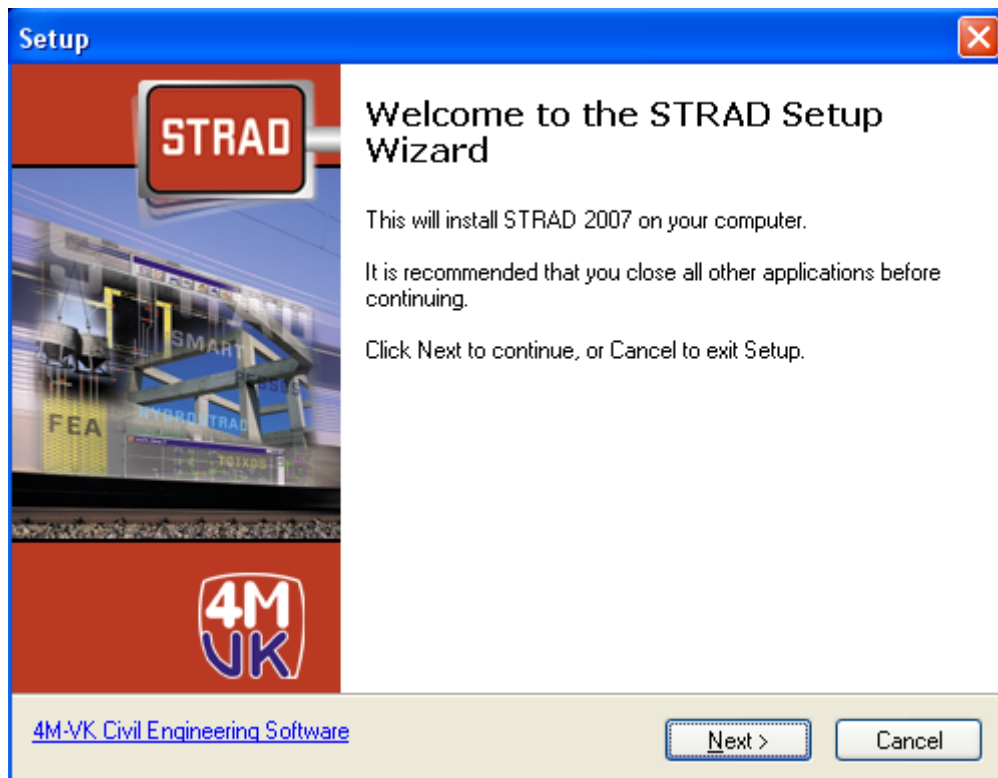
---

# 1. Installation - Launching

---

## 1.1 Installing STRAD

1. Insert the CD in your computer CD-ROM drive (e.g. D:, E:) or, if you received your software via Internet, run the installation application you downloaded.
2. When the Setup window appears, choose the language for the installation and click OK.
3. When the Welcome page appears (as shown below), click **Next**.



4. Follow the on-screen instructions until the installation is completed.
5. After installation, the program is located within the programs list.



---

## 2. Working with STRAD

---

### 2.1 Introduction

STRAD is an analysis program using linear elements and designing structures from reinforced concrete. Optionally, it supports analysis using Plane Finite Elements and designing structures from steel, Load Bearing Wall and timber structures. In particular, It includes the following parts:

Description of the structure: The program is based on IntelliCAD, which provides an Auto-CAD like graphic interface to “describe” the structure easily and quickly.

Analysis: The program include Analysis by Modelling all elements as linear members. For modelling shear walls and slabs as Plane Finite Elements, a purchase of an additional program of 4M-VK is required.

Design: With STRAD you can design reinforced concrete members. Optionally, you can design Load Bearing Wall, plain or with metal or timber headsills.

Report Generator: Description of results, printings, technical report etc

### 2.2 Terminology

Space frame analysis becomes using the solution of the equations  $[R]=[K].[r]$ , where:

$[R]$  = Load Matrix

$[K]$  = Resistance Matrix

$[r]$  = Deformation Matrix

In order to solve this equations the program needs the following information:

- Coordinates for each node, that is to say X,Y,Z.
- Members connectivity: start and end node for members.

- Member properties: inertia moments and area, Modulus of Elasticity.

With these information, the Resistance Matrix [K], can be created.

Bars and nodes, of which their properties are used for Matrix K creation, will be called static entities.

Finally, for Load Matrix [R] creation, nodal forces are required (up to 3 loads and 3 moments in each node for each Load case).

Data import of this form would be very difficult, for a structure. The program < < recognizes > > "floors, beams, slabs, columns, shear walls, foundation in a local system (plan). Also, it accepts slab loads, which automatically transports in the beams. Regarding to seismic loads, it is enough to give the seismic factor and the way of earthquake distribution (eg triangular load).

These entities are changed automatically in files of the global coordinate system, that is to say in a space frame model. Space frame model (DATA files) can be changed as: inertia changes, increase of seismic load in some level (eg pilotis) etc.

Then the equations matrix are created and solved. At first, displacements and torsions of nodes are computing. From these results Members Internal Forces and Moments are estimated. These are presented analytically for each Load case.

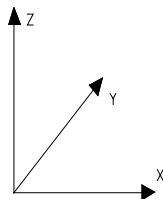
Members design becomes from the worst combination of Internal Forces and Moments, based on the code that you have selected.

This is a simple description of **STRAD**.

Before you go on reading the program operation, study carefully the rest chapter, in which, there is an analytical explanation of the terminology that is used.

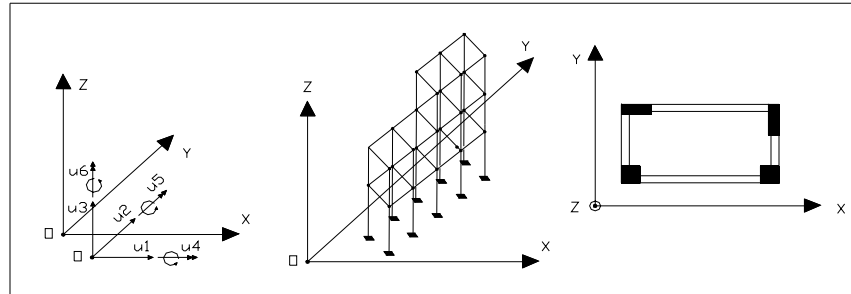
### 2.2.1 GLOBAL COORDINATE SYSTEM

For a structure description in the space, coordinates X,Y, and Z are necessary. In STRAD all coordinates, component vectors due to loads and displacements are described in right handed coordinate systems. Such a coordinate system appears in the following figure:



where X,Y,Z, are the axis directions and u1 up to u6 are the six component vectors due to a load or a displacement.

Global coordinate system is fixed as the above figure shows in order that X axis is horizontal and plan description take place in a plane parallel to OXY.



**Warning:** Plane frame

s should be placed in a plane parallel to OZY of the 3D coordinate system (That is to say X coordinate is constant)

STRAD solves structures that constituted by linear members which aid in nodes, in which acting forces. This is, to describe these members in a KNOWN working «Local system», that is to say to recognize: columns – beams – footings – strip footings – slabs etc.

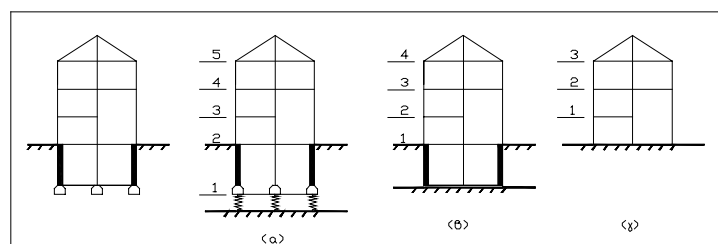
### 2.2.2 LEVEL

Level is a plane which intersects the vertical structural members, such as columns or footings. Slabs and beams belongs to a level. In most structures, level is the slab of a floor or the foundation, but naturally, can be also anything that split the continuity of vertical structural members.

Numbering begins from the lower level that you want to describe, by giving number 1 and increasing successively until the last level. In case you want to solve the structure with the foundation, level 1 will be the foundation level (that is to say footings - soil interface).

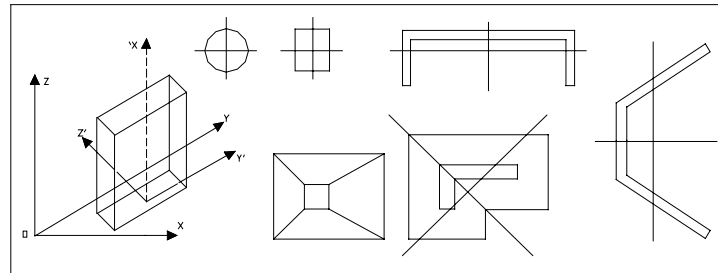
Program automatically creates also level 0, which is the fixed level (displacements and torsion are minor). In case where foundation is described, level 0 is the simulation plane of subsoil. Elevation of level 0, is  $\pm 0.00$ . In the program we cannot define negatively elevations. Levels are parallel to plane OXY and have ELEVATION +Z(m). For each level should be defined an initial elevation, too. We call it initial, because we can change level elevation partly, afterwards. In this case, you should give the elevation in which there are most of the columns, so that it will need less corrections, afterwards. The elevation of each level is the absolute altitude from level 0 (the program arbitrarily considers that the elevation of level 0 is zero). If level 1 is the foundation, then this level will arbitrarily, have elevation 1m.

In the following figure there are given 3 likely descriptions of the same building. In (a) there is a description of entire the building with the foundation. In (b) building is considered as fixed in the foundation, while in (c) it is considered as fixed in the ceiling of the basement.



### 2.2.3 NODES (COLUMNS, SHEAR WALLS, FOOTINGS)

Column (or shear wall or footing) is a vertical member that starts at the fixed level (0) and connects successive levels. It can be rectangular, circular or complex, constituted by rectangular sections (up to 7 sections), saped as it appears in the following figure. It is a linear member and it is modelling in its centroidal axis.

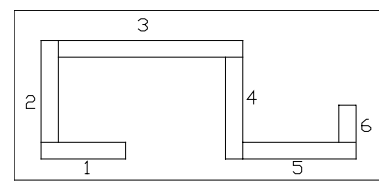


**Note:** Suspension column does not belong in this category because it does not start from level 0.

Node is the intersection of column axis to some level and it is described by coordinates X, Y, Z (in cm). X, Y are the coordinates of c.g. (they are described straight from the user or calculated by the program in the case of complex column or a description of fixed point) and Z is the level elevation.

The column numbering is the same in all levels (that is to say, each column has the same number in every level).

Section numbering of complex cross-sections becomes by a decimal number, that the first digit is referring to the cross-section (the number of the node) and it is the same for all sections (which constitute the complex cross-section) and the second digit is referring to the section of complex cross-section. In order to achieve in CAD a better design of complex cross-section, you should number the sections so that section n+1 follows section n.



### 2.2.4 DIMENSIONS

Rectangular shaped columns or sections of complex cross-section have dimensions B (dx) and D (dy). Where, dx is the column side that initially is parallel to OX axis of the global system and dy is parallel to OY axis. We say initially, because we can change the angle, as it is described below.

### 2.2.5 COORDINATES

Columns coordinates X,Y are defined in the global coordinate system OXY. Coordinates cannot be negative. It is recommended min values for X,Y, equal to 4m. This happens because Plates and VKCAD (design programs) will not <<recognize>> lines with negative coordinates (consider cantilevers and footings). This is why (4,4) provides for a cantilever length up to 3m and footing with cantilever up to 3m (to their axis OX, OY). If you want to create bigger cantilevers or footings then you must use respectively greater coordinates. The program reduces coordinates by the c.g. of cross-section (complex or simple) and fixes the altitude Z of each node equal to the level altitude.

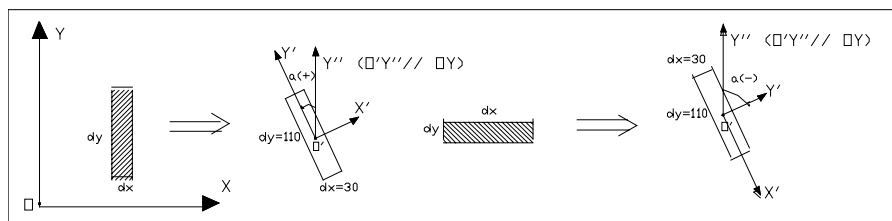
### 2.2.6 ROTATION ANGLE

Rotation angle of canted columns or sections of complex cross-section is the angle between the side dy (final location O'Y') and the axis OY of the global coordinate system, parallel displaced at the rotation point (be it O'Y''). Rotation angle is the angle between O'Y'' and the final place of dy O'Y' and it is positive for anticlockwise direction. Rotation axis goes through the point in which coordinates were given (c.g. or fixed point).

Sides and rotation angle have direct dependence between them. As you see in the next figure you can achieve the same result by changing dy/dx and rotation angle. Here it is recommended that you must not use angle greater than 45° in absolute values (rotation angle can be negative)

### 2.2.7 CIRCULAR COLUMNS

Circular columns are numbered regularly with the rest columns but they are only specified by the coordinates of c.g. and for dimension it should be given radius in cm.



### 2.2.8 FREE NODES or just NODES

Free nodes are all the nodes which do not belong to the previous category (intersection of column or footing with level). It can be the free edge of a cantilever, the point that supports a beam on a beam, the point that supports a suspension column on a beam etc.

Their description becomes per level, their NUMBERING does not depend on column numeration at the same level and it is independent from level to level.

Their COORDINATES are in the global coordinate system OXY and they are given to each level. As x,y eg in the case of beam on beam it is given the point of intersection of axes of the two beams.

**Warning!** A usual error is the description of a free node which is not connected to the structure. Each free node should always be connected to some member of the structure.

### 2.2.9 BEAMS

Beams are the linear members which connect 2 nodes (column faces or free nodes) of the same level.

Beams are numbered independent at each level. It is recommended, to keep the same numbering if there are typical levels so that you can take advantage of copy possibility.

### 2.2.10 DIMENSIONS

Beams are considered that they have rectangular web of dimensions BxD either they are T-shaped or not, where B= width and D=total height including slab thickness.

### 2.2.11 SHAPE

Additionally by cross-section dimensions, should be also specified the shape of beam. For description of beams in plans, there is a "shape" library of beams (shape 0, L, T, Z, L, inverted T, inverted L). **Mark that, depending on the shape of beam the program will decide if there is stiffness lengthwise.**

Specifically, if the shape is other than 0 (simple rectangular cross-section) then it will be considered that there is a great level stiffness (it is given  $Jz/L=1 \text{ m}^4$  - see DATAM and «Members Local System»).

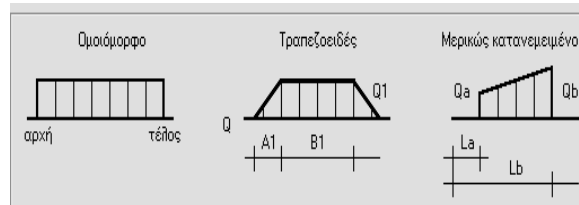
Also, by the shape and slab thickness the effective width will be calculated.

### 2.2.12 CONNECTIVITY (START NODE – END NODE)

The two beam edges (Start, End) are specified by two nodes which can be columns, rectangular sections of complex cross-section or free nodes.

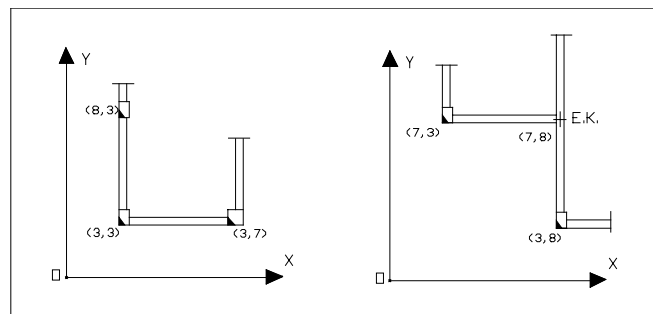
### 2.2.13 BEAM LOADS

Beam load is given in t/m (for allowable stress method) and KN/m (for N.G.R.C.C.) and can be uniformly, trapezoidal or triangular.



- The uniform load is symbolized as  $Q$
- Trapezoidal load is symbolized as  $Q1$
- Lengths are shown in the figure and they are in m.

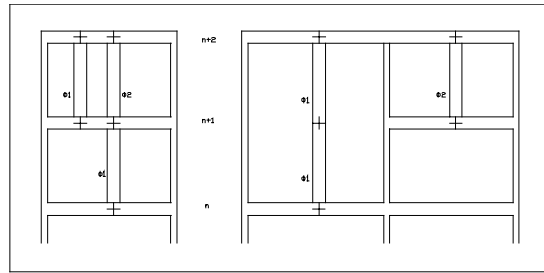
By zeroing  $B1$ , you can describe the triangular load. In order to give an axial load acting in particular spot of a beam, you should <<give>> at this place a free node.



In case where the beam loads are transferred automatically by slab analysis, then they are uniformly in t/m or KN/m depending on the code.

### 2.2.14 SUSPENSION COLUMNS

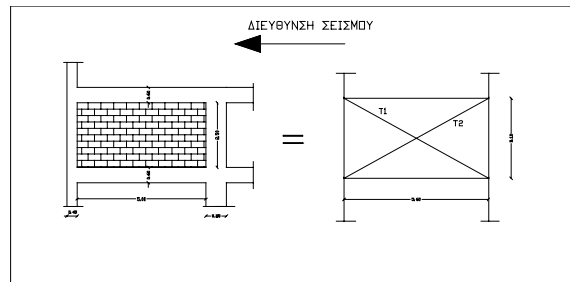
Suspension columns are the vertical members which do not have fixed ends at level 0. They might have a rectangular or circular cross-section but not complex. Suspension columns are described in the level where they «end» and they are «on» the immediately previous level.



Their NUMERATION is independent for each level and their dimensions are in centimetres. They are described like rectangular and circular columns and they can also have a rotation angle. Suspension columns should be on free nodes which should have been inserted at levels. Start node is the foot node of the suspension column, that is to say the free node of the previous level. End node is the free node at the top of the suspension column, that is to say the level at which we describe the suspension column. Free nodes coordinates X and Y of foot and top should be the same.

### 2.2.15 INFILL WALL (WALLS)

This is modelling of inbond walls, without openings, infill walls. This members are diagonal between frames in the places the above walls exist (see the following picture) and function as compressive members (pinned at both ends) in the antiseismic model. Infill wall members are given at the level where walls are 'stepping' on and connect columns gravity centres or free nodes of the same level to the next. Their NUMERATION is independent in each level.



### 2.2.16 CONNECTIVITY

Definition of FOOT/TOP LEVEL and FOOT/TOP NODE is necessary at this part. Foot level is the level in which infill wall is supported and cannot be the zero level. Top level is the immediately next level. This level is also the level at which we describe infill walls. As well, foot node is the connection node at foot level and the top node the node where the member ends at the immediately next level.

### 2.2.17 DIMENSIONS

For dimensions it could be given member thickness, which corresponds to the real thickness of inbond wall, and a value for its height. Wall height does not coincide with the real height of infill wall. Wall height depend on parameters that are relating to the structure eg frame dimensions, stiffness and building height. An average volume is 80-100 cm.

**Note:** Walls as well as suspension columns are faced as auxiliary members by the program. (See below).

**Warning!** Infill wall members are very useful for the reduction of displacements due to seismic forces. Their use is not recommended in projects of new buildings where you cannot be sure for the place and situation of infill wall. You can, however, use them in projects for existing structures.

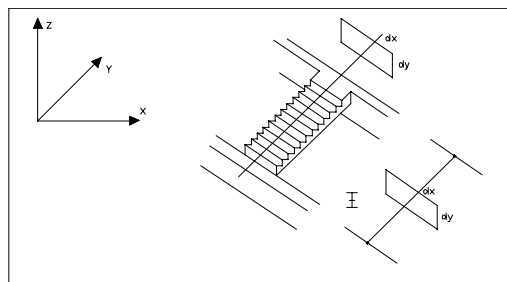
### 2.2.18 AUXILIARY MEMBERS

Auxiliary members are the members of the structure that do not belong in the categories of footings, beams or columns (eg stairs, steel members etc). Auxiliary members can have any cross-section shape, be constituted from any material and connect two nodes that belong in the same level or in different levels. Also they can connect gravity centres of columns and not faces as beams do. They cannot start from level 0. Auxiliary members are usually used for the description of steel members, stairwells and in connectivity of perimetric basement shear walls. Their NUMERATION becomes for all the building, does not dependent on level and place. From the numeration are excluded auxiliary members of perimetric basement shear walls, which are given in other location.

### 2.2.19 DIMENSIONS

Auxiliary members can be described in two diferent way:

- a. If they are rectangular shaped, by giving the sides width and height in centimeters. If they are vertical elements then the width is parallel to OX axis of local system. We can give also a rotation angle, as we do in columns. For horizontal or sidelong elements in space (eg stairs), width is perpendicular in the plane of level or stairs, as it is shown in the following figure. Rotation angle is the rotation of member axis according to the main axes (sidelong column has a rotation angle, while stairs does not have).



- b. By typing moments of inertia  $J_x$ ,  $J_y$ ,  $J_z$ , in decimetres in fourth ( $\text{dm}^4$ ), area  $A$  ( $\text{cm}^2$ ) and rotation angle (if there is).  $J_x$  is the torsional moment of inertia and  $J_y$ ,  $J_z$  are the bending moments which are fixed as  $dy$ ,  $dx$ , respectively (see also «Members Local Coordinate System»).

### 2.2.20 CONNECTIVITY

Because, auxiliary members can link nodes in different levels, their connectivity is fixed by START LEVEL and END LEVEL.

Start Level and End level cannot be level 0, but they can be the same or different levels.

Also, START NODES and END NODES are required. Start node is the start level node of auxiliary member.

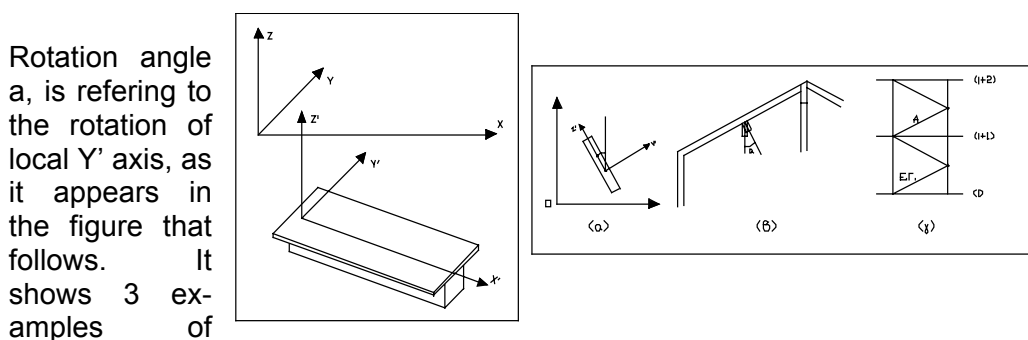
Auxiliary members have no dimensions.

### 2.2.21 MEMBER LOCAL COORDINATE SYSTEM

Moments of inertia  $J_x$ ,  $J_y$ ,  $J_z$ , are referring to the member local coordinate system. Actually, this system is the global coordinate system (dextral cartesian) applied at the start node of members. Local X axis is the same with the member axis and direction is positive from member start to its end.

For columns,  $Y'$  axis is parallel to side  $dy$  and  $Z'$  is parallel to  $dx$ . When a column is not side-long then the axes are parallel to global coordinate system.

For beams,  $Y'$  axis is parallel to the level and  $Z'$  is perpendicular to the level.



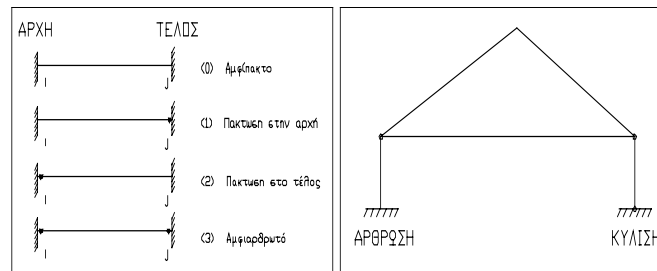
member rotation. The first is the classic sidelong column that has already been described. The second concerns a beam of an inclined roof which initially is vertical in the plane of level (which is under inclination). Rotation angle is  $+a$ , if X axis «comes» (it is vertical) from the plane of the paper. In the third figure it is shown a stairwell (auxiliary member) in which however  $a=0$  (it does not have rotation angle). The angle that is created at Z because of the difference of S/N and E/N is calculated by the program, but it is not the rotation angle  $a$ .

**Note:** The local system is also applied for internal forces and moments of results.

## 2.2.22 MEMBER PINNED

A possibility that you have by using STRAD is to define pinned end in member edges, that is to say a rotation possibility in the direction of two bending moments but not in torsion.

The program considers that all members of structure (except wall members) are fixed at both ends. You can describe a pinned end at the edge by giving other support type, as: Member fixed at both ends, Member fixed at start node and pinned at end node, Member fixed at end node and pinned at start node, Member pinned at both ends.



In order to achieve a rotation possibility or movable (whithout a rotation possibility) in supports you apply the suitable modelling as it appears above, that is to say the pinned with a bar fixed at the end node and pinned at the top node and movable (whithout a rotation possibility) with a bar pinned at both ends.

## 2.2.23 SHEAR FACTORS

STRAD also, considers deformations due to shear stresses.

$$\text{Shear deformation } \mathbf{Y} = \frac{\mathbf{V}}{\mathbf{k} \cdot \mathbf{F} \cdot \mathbf{G}}$$

- where
- $\mathbf{V}$  = Shear force
  - $\mathbf{F}$  = Total cross section area
  - $\mathbf{G}$  = Material shear modulus
  - $\mathbf{k}$  = Shear factor

Shear factor  $k$ , is the percentage(%) section of cross-section that undertakes shear and depends on cross-section shape. By decreasing  $k$ , shear deformation is increasing and bending deformation is decreasing.

For rectangular sections  $k=0.8$ . For other shapes see « Technica Hronika» No 7-9/85 p. 45.

#### 2.2.24 FOUNDATION

As we have already mentioned, you can also describe the foundation level for total analysis. Foundation can be constituted by footings and grade beams, strip footings or even mixed foundation. Description of foundation level is the same as the other levels with some differences that we will mention below.

#### 2.2.25 FOOTINGS & GRADE BEAMS

Footings are faced as columns according to their numeration, dimensions, rotation angle and coordinates (circular footing cannot exist).

Footing dimensions are in centimetres. Coordinates are given by the ordinary way for each rectangular footing or footing section. You can also found a complex column of level 2 in a simple rectangular footing.

Grade beams usually have shape 0 (simple rectangular cross-section).

#### 2.2.26 STRIP FOOTINGS

When you describe strip footings in foundation level you should have in mind the following:

**Shape.** Usually strip footing shape is rectangular, L-shaped or inverted T.

**Dimensions.** As well as in beams you must give web dimensions. Moreover, for shapes L or inverted T, you should also give «slab thickness», in order to calculate «effective width». In the position «slab thickness» you give the thickness of effective width and for «effective width» the width of effective width.

**Connectivity.** It is defined by start/end node. Columns and free nodes of foundation level are considered as nodes. In case of mixed foundation, a footing cannot be given as start/end node. Only a grade beam can link a footing to a strip footing (start or end) node (usually it is a free node in the interface of footing – strip footing, that is connected with the gravity centre of footing by a auxiliary member with great rigidity). This is happening, because in footings, in which aid strip footings, inertia elements are zero.

**Columns dimensions which connects to strip footings.** Apart from free nodes dimensions you must describe the dimensions of strip footing connection nodes, too.

Here it accomodates to give the column dimensions which continue (usually columns at level 2 are copied).

#### 2.2.27 SPACE FRAME

Space frame model is the structure that becomes from the data that we have described as a single space frame.

**DATAM** :Table of connectivity for members, moments of inertia, area and modulus of elasticity.

**DATAK** :Table of node coordinates, in global coordinate system.

**DATAKM** :Table of member face coordinates (start and end), in global system.

**DATAF** :Table of loads.

### 2.2.28 DATAM

For each member it is calculated connectivity, moments of inertia, modulus of elasticity, shear modulus and Euler angle. These values can be modified as changes in DATAM. We give the corrections - new values in the second line under the value that we want to change. The elements of the second line constitute the **DATAM of changes**. From DATAM (Initial and Changes), becomes the **Combination DATAM** from which the analysis will become.

It is pointed out that the Combination DATAM does not present in the screen, but only the two others.

Usual changes in DATAM become for the following reasons:

- a) To change stiffness of some member by moments of inertia fluctuation.
- b) Description of other material by changing M/E (eg steel, timber).

Also we should not use zero value for area, moments of inertia or M/E of some member. If we want however, to give very small value for the area or moments of inertia, we type the value 0.00001.

#### Symbolization

JZ	:Moment of inertia in member local Z axis, in m <sup>4</sup> .
JY	:Moment of inertia in member local Y axis, in m <sup>4</sup> .
JX	:Moment of inertia in member local X axis, in m <sup>4</sup> .
F	:Area of member cross-section, in m <sup>2</sup> .
a	:Rotation of main axes Y, Z of the member local system in degrees.
E	:Material Elasticity modulus.
g	:Shear factor, to calculate shear modulus G,

where  $G = g \times M/E$ . For reinforced concrete  $g=0.40$ .

## 2.2.29 DATAK

### Generally

In DATAK there are coordinates of all nodes (columns and free nodes) in the global system. Changes becomes directly in the graphic area. Columns coordinates are defined from the top. Only in level 0 they can be defined by the bottom.

**Note !!!** If we have a split level foundation and we “raise” some nodes of 1st level from 1.0m, to 3.0m (change = + 2), then we should “raise” level 0 of those nodes for 2.0m, as well.

**Warning !!!** Changes which are made at Z axis, should be such that they will not nullify the length of some member. Also, we should be careful, that we do not create negative member with “negative” length, that is to say we should not raise a column start node higher than the end node.

### Symbolization

X :X node coordinate of the global system, in m.  
Y :Y node coordinate of the global system, in m.  
Z :Z node coordinate of the global system, in m.

## 2.2.30 DATAKM

Members that STRAD is using, can be deformed only in one of their section (fixed node at both ends). Therefore, in DATAKM file there are node coordinates i, j (real member start and end).

At first stage, for columns, are the same as CG coordinates that we have given at the same level and at the next one. If there is a displacement in CG of the bottom level then it is created a constant node. For beams (not strip footings), the deformable way is defined at the length of axis of 2 foreheads of beams at the level altitude.

### Symbolization

Xst.(m) :X member start node coordinate of the global system, in m.  
Yst.(m) :Y member start node coordinate of the global system, in m.  
Zst.(m) :Z member start node coordinate of the global system, in m.  
Xen.(m) :X member end node coordinate of the global system, in m.

Yen.(m) :Y member end node coordinate of the global system, in m.

Zen.(m) :Z member end node coordinate of the global system, in m.

### 2.2.31 DATAF

This is the file of nodal forces. Beam distributed loads are analyzed in nodal forces. All loads are referring to the global system.

#### Symbolization

Dead(LC1) :Nodal force due to vertical dead loads.

Earthquake Y(LC2) :Nodal force due to earthquake at Y-Y.

Earthquake X(LC3) :Nodal force due to earthquake at X-X.

DT1, DT2,  $\Delta\theta_1$ ,  $\Delta\theta_2$ :. Nodal force due to temperature changes, that is imposed in the beams. 4 load files are created (4,5,6,7) for loads DT1, DT2, DC1 and DC2 respectively.

LIVE(LC8) :Nodal force due to vertical live loads.

LC9 :Load case which can be defined from the user.

LC10 :Load case which can be defined from the user.

LC 11 :Load case which can be defined from the user or vertical seismic load if there is use of GAC2000.

Note!!! Apart from the above 11 load cases, there is L.C.12 which is created when we ask from the program to make load combinations for beams. This becomes in code "ANALYSIS". The only difference is that we cannot "see" L.C.12, in the contrary to the other 11 load cases.

xFX :Multiplication factor for the parallel to X-X level load.

xFY :Multiplication factor for the parallel to Y-Y level load.

xFZ :Multiplication factor for the parallel to Z-Z level load.

FX :Nodal force parallel to X axis of the global system, in T or in KN.

FY :Nodal force parallel to Y axis of the global system, in T or in KN.

FZ :Nodal force parallel to Z axis of the global system, in T or in KN.

MX :Start nodal moment, which has vector parallel to X axis of the global system, in TM or KNM.

MY :Start nodal moment, which has vector parallel to Y axis of the global system, in TM or KNM.

MZ :Start nodal moment, which has vector parallel to Z axis of the global system, in TM or KNM.

DT1: Axially imposed temperature change, signed number. (Dilatation (+) and Shrinkage (-)). It is in °C.

DT2 :Axially imposed temperature change, signed number. (Dilatation (+) and Shrinkage (-)). It is in °C.

$\Delta\theta_1$  :.Crossly imposed temperature change. (Top and bottom fiber). It is in °C.

$\Delta\theta_2$  :.Crossly imposed temperature change. (Top and bottom fiber). It is in °C.

$\alpha$  :Coefficient of thermal Expansion.

By concluding, all STRAD entities are being handled as IntelliCAD objects. In particular, the drawing objects that are related to STRAD are levels (layers), column, suspension column, node, beam, wall, auxiliary member, footing, strip footing, cantilever line, hole line, Reinforced Slab Strip line, Analysis Strip line, slab. All this information is contained within a DWG type of file.

#### LIMITATION OF STATIC ENTITIES

Levels	27
Columns /level	500
Free nodes /level	1000
Additional slab points /level	2000
Beams /level	500
Auxiliary Members	1000
Auxiliary Members at one level + Beams /level	500
Walls /level	500
Basement shear walls + columns /level	500
Minimum coordinate (suggested)	5m
Minimum coordinate	0m
Maximum coordinate (suggested)	320m

Maximum eigenmode number for linear members	19
Minimum moment of inertia, area, A	10 <sup>-6</sup>
Maximum J/L (suggested)	100
Maximum K for soil (suggested)	10000000
Minimum member length (suggested)	0.1m
Maximum slab number	100
Maximum number for Analysis Strips per type	50
Maximum beam load	32000

## 2.3 Example

This chapter describes a quick example that helps the user to realize STRAD's basic concept and functioning through a step-by-step procedure.

### NORMAL OPERATION

#### 1<sup>st</sup> STEP: Start the program. New Project.

S/No	Secondary steps	Analytical instructions
1	<b>Start the program.</b>	<p>To start the program perform one of the following:</p> <p>From the basic Window menu select START&gt;PROGRAMS&gt;STRAD 2001&gt;AUTOSTRAD 2001,</p> <p>From the working surface of the Windows double click on the shortcut.</p>

Wait until the starting operation of AutoCAD 14/2000 is complete. You can establish that in the command line of AutoCAD 14/2000 when the prompting 'command' appears.

## 2 New Project

To create a file of a new project perform one of the following:

Select the **FILE>NEW STRAD PROJECT** command.

LC on the icon  in the toolbar.

The dialogue box for the NEW STRAD PROJECT 2000 appears. Accept the number '100' as the "Three-Figure Project Number". With this number you can name the file, which includes this project's files.

In the field "Number of Levels" type the number '3'.

Make sure that the "Safe Mode" is unmarked.

S/No	Secondary steps	Analytical Instructions
		<p>Optional, you can type the rest of the sections that you consider as necessary. Those sections help you distinguish your project from other projects in the saving projects' catalogue.</p> <p>When you complete the input press [OK] to close the dialogue box.</p> <p>In the command line the program asks {Changes of the existing project will be lost. Continue?} Press [OK].</p> <p>The program automatically opens the dialogue box LEVEL PROPERTIES and leads you to the next step, the specification of the level properties.</p>

## 2<sup>nd</sup> STEP: Level Properties

S/No	Secondary steps	Analytical Instructions
3	<b>Level Properties.</b>	The dialogue box LEVEL PROPERTIES should be open from the previous step. The initial picture of the dialogue box should be the same as the one of Figure 1.

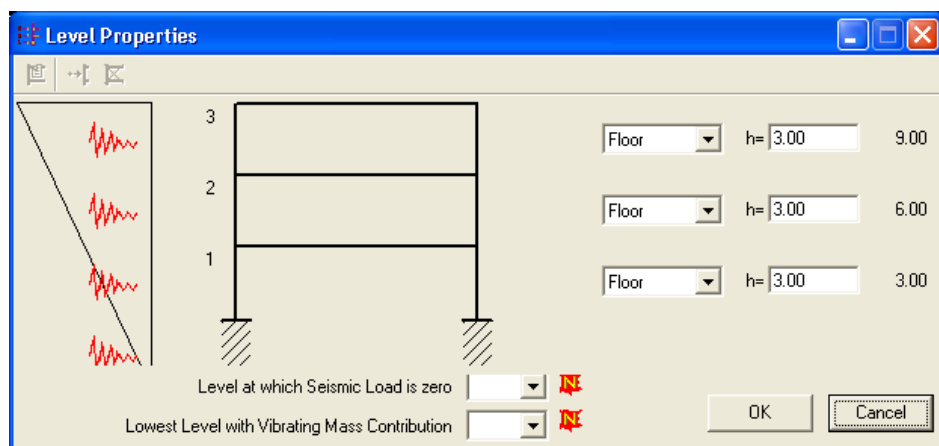


Figure 1 The initial form of the LEVELS frame.

S/No	Secondary steps	Analytical Instructions
4	<b>Level Specification.</b>	<p>Start by specifying the levels.</p> <p>LC to open the list on the right of the first - from the bottom - level.</p> <p>Select 'Foundation'.</p> <p>LC to open the list on the right of the second - from the bottom - level.</p> <p>Select 'Basement'.</p> <p>At the third level leave the specification 'Floor'.</p>

Move on to the altitudes.

Type in the corresponding - for the basement - box the value '4'.

Type in the corresponding - for the third level - box the value '3'.

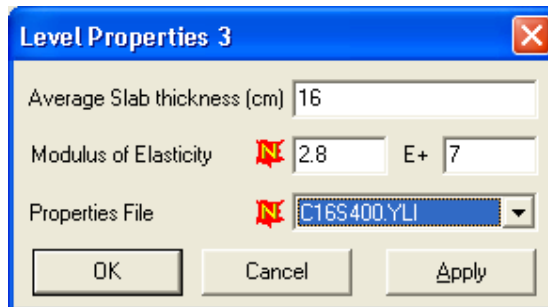


Figure 2 The dialogue box Level Properties 3.

Specify the appropriate Properties File.

Inside the rectangular that sets the third level, RC on the drawing. A dialogue box appears and LC on the 'Level Properties'.

S/No	Secondary steps	Analytical Instruction
------	-----------------	------------------------

Figure 2 appears.

Specify the Properties File.

Inside the rectangular, which sets the third level, RC on the drawing. The dialogue box of Figure 2 appears.

In the box 'Average Slab Thickness' type the value '16'. In the box 'Modulus of Elasticity' leave the value as it is.

From the list 'Properties File' select the C16S400.YLI.

Press [OK].

In the level 'Basement' the program sets by itself the properties of Level 3. Allow it as it is.

In the foundation level repeat the above procedure by setting the value 0 in the 'Average Slab thickness' and select the C16D400.YLI from the Properties File list.

When you finish this press [OK] to close the window.

In the dialogue box Levels perform the following:

In the box 'Level at which Seismic Load is zero' select the value '2'.

In the box 'Lowest level with Vibrating Mass Contribution' select the value '3'.

In the Figure 3 you can see the dialogue box as it is after the commands completion.

Press [OK] to close the dialogue box.

## **5 Save your project**

From the **FILE** menu of AutoCAD 14 select **SAVE**.

Saving your project with the AutoSTRAD 2001 is a process that you should repeat very often during this project analysis.

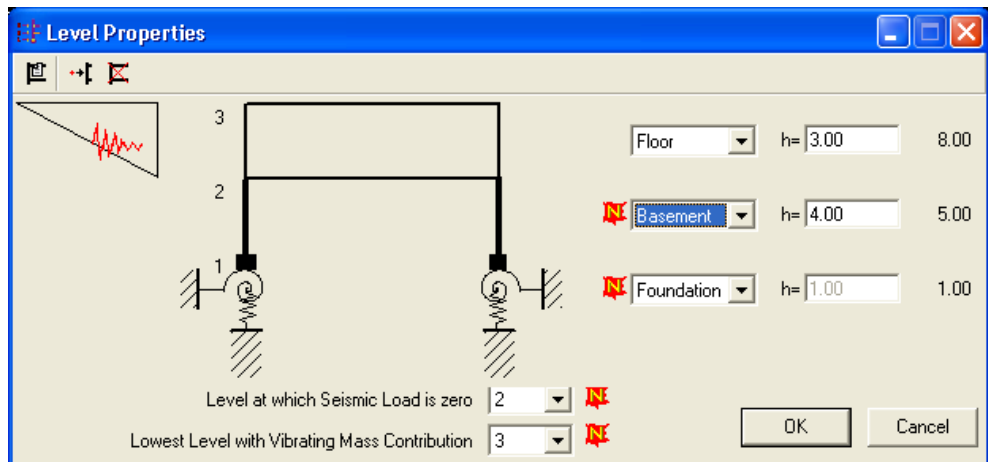


Figure 3 The Level Properties after the commands' completion.

### 3<sup>rd</sup> STEP: The preparation for the specification of the structural system.

S/No	Secondary Steps	Analytical Instructions
6	<b>Rectangular Shaped Columns</b>	<p>In order to predetermine the characteristics of the rectangular column's sections process as followed:</p> <p>Select the <b>MODEL&gt;DEFAULT MEMBER PROPERTIES&gt;RECTANGULAR SHAPED DIMENSIONS</b> command. The dialogue box of the Figure 4 appears.</p> <p>In both lists select the value '0.50'. Or you can type them in the boxes I1 and w1.</p> <p>For a fixed point select the value '0'.</p> <p>Press [OK] when you finish this.</p>

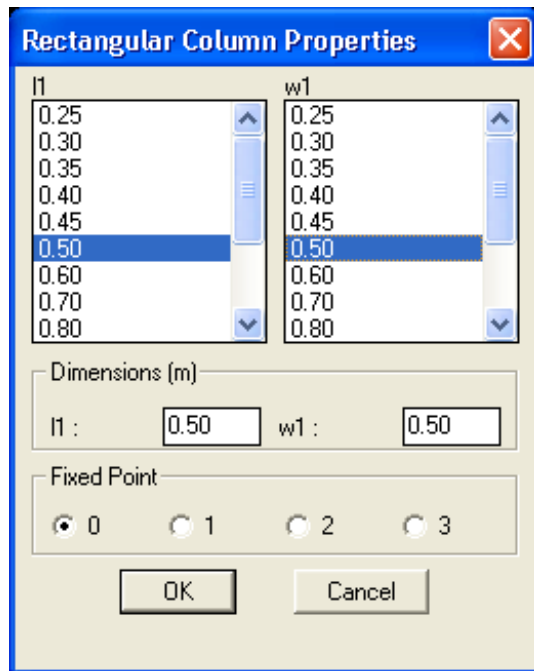


Figure 4 The dialogue box for the characteristics of a Rectangular Shaped Column.

S/No	Secondary steps	Analytical Instructions
7	<b>Core</b>	<p>To specify the characteristics of the core section follow the process:</p> <p>Select the <b>MODEL&gt;DEFAULT MEMBER PROPERTIES&gt;CORE SHAPED DIMENSIONS</b> command. The dialogue box of the Figure 5 appears.</p> <p>In the boxes set the following values:</p> <p>In 'L1' the value '2.40'</p> <p>In 'L2' the value '2.00'</p> <p>In 'L3' the value '2.40'</p> <p>In 'w' the value '0.25'</p> <p>Press [OK] when you finish this.</p>

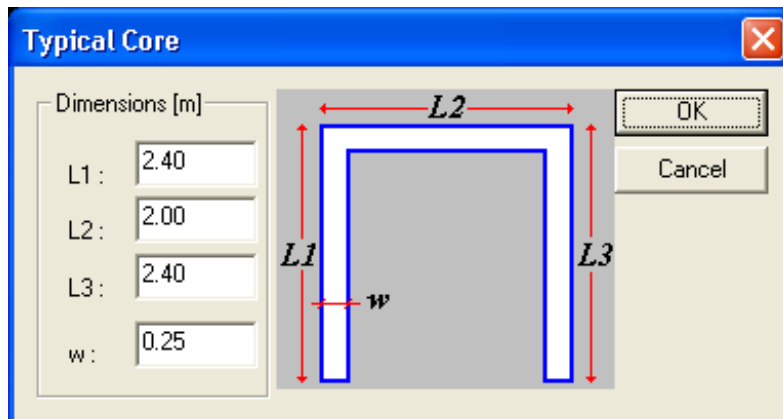


Figure 5 The dialogue box with the characteristics of the core.

S/No	Secondary steps	Analytical Instructions
8	<b>Beam</b>	<p>In order to specify the characteristics of the beams' sections follow the process:</p> <p>Select the <b>MODEL&gt;DEFAULT MEMBER PROPERTIES&gt;BEAMS</b> command. The dialogue box of the Figure 6 appears.</p> <p>Select by LC the card 'GEOMETRICAL DATA' where you can apply the following changes:</p> <p>Select the 'Γ' shape as a section type.</p> <p>In the box 'Height' type the value '0.60'.</p> <p>In the box Width' type the value '0.25'.</p> <p>Leave the rest of the boxes without making changes and press [OK] when you finish this.</p>

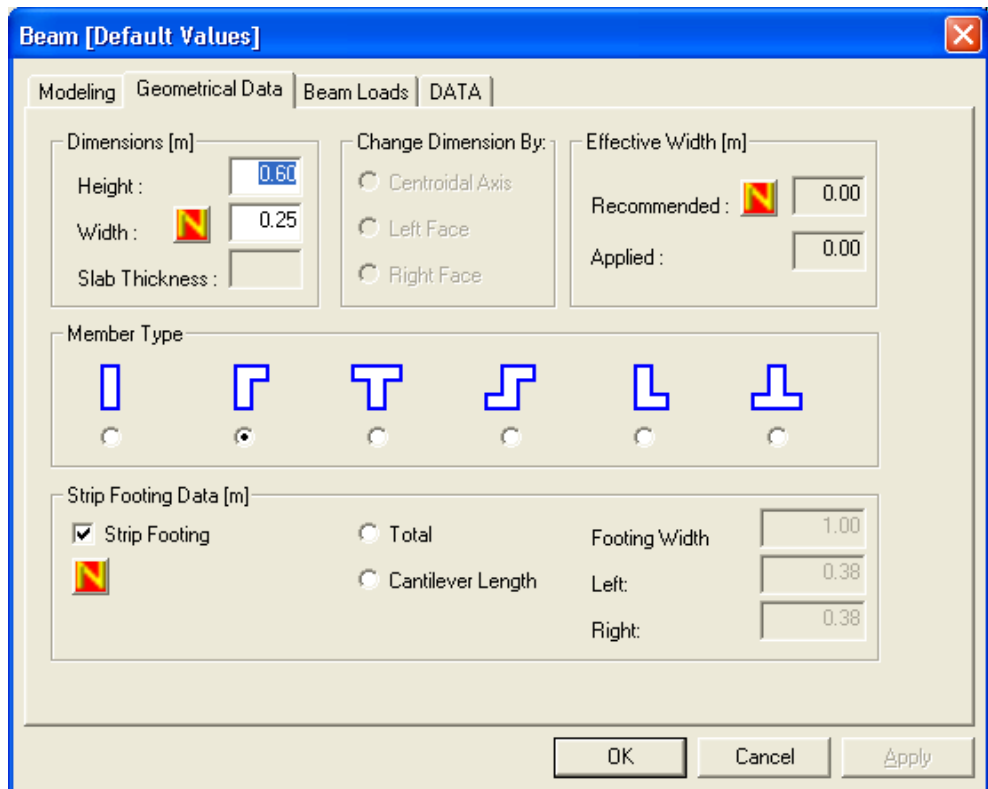


Figure 6 The dialogue box with the Geometrical data of the beams.

## 4<sup>TH</sup> STEP: PROPERTIES FILE: Specification of the parameters.

S/No	Secondary steps	Analytical Instructions
9	<b>General Parameters</b>	<p>If you wish to open both the Properties Files that you have specified at the second step and check the parameters, follow the process below:</p> <p>Select the <b>ANALYSIS-DESIGN&gt;GENERAL PARAMETERS</b> command. The dialogue box of Figure 7 appears.</p> <p>Open the list by pressing the arrow and select the file E16S400.YLI that has been set for the ROOF and the BASEMENT level.</p> <p>Press [OK].</p>

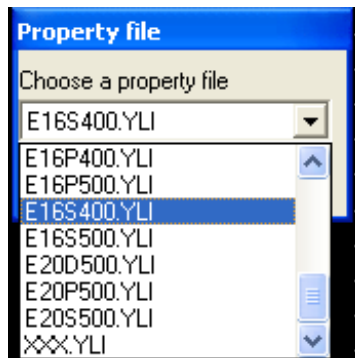


Figure 7 The Properties File dialogue box.

**10 The file  
E16S400.YLI**

The dialogue box of Figure 8 appears. All the parameters are organized in units - cards. You can move from one to the other by choosing its title.

With an analogous way the parameters of the File E16S400.YLI, set for the Foundation Level, appear.

Some of the parameters, like the ones for the Seismic Code, are effective for the whole construction and thus you only need to specify them in the Properties File E16S400.YLI. The program applies them in the files of the project (in our case in the E16S400.YLI, for the foundation). On the contrary the parameters that are specified for a level (like the soil elements) are taken from the Properties File, which is specified for this level, while the corresponding values of the rest are ignored.

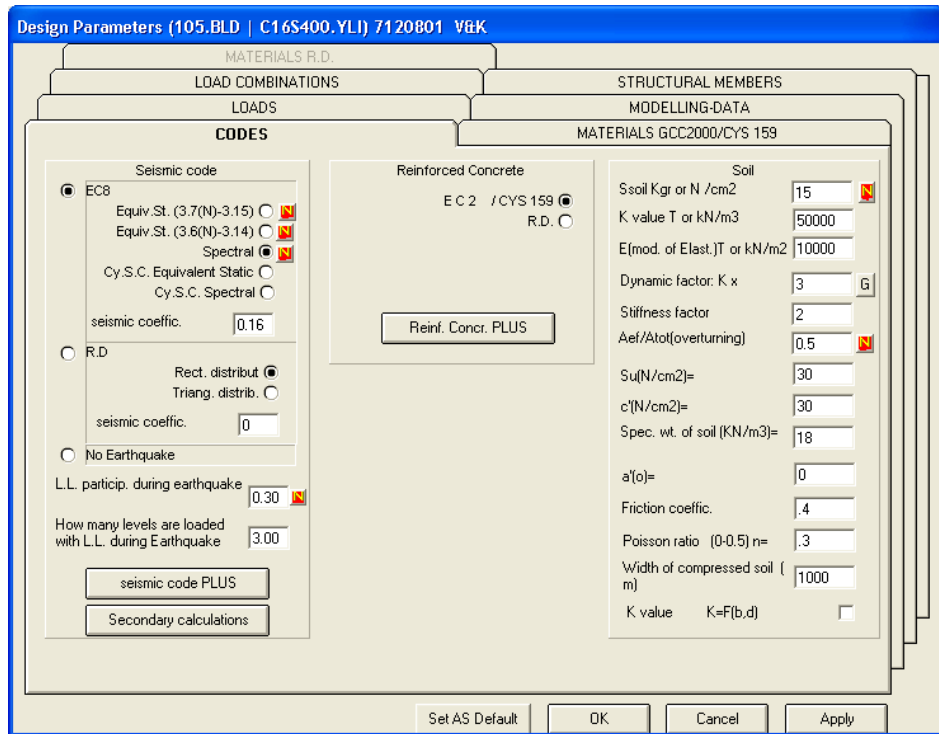


Figure 8 The DESIGN PARAMETERS dialogue box.

**11 Parameters Seismic Code**

If it has not appeared you can LC on the card 'CODES' and perform the following:

Select "EC 8" and LC on the box 'seismic coefficient'.

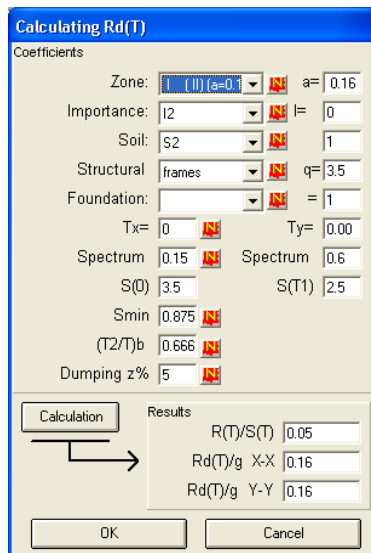





Figure 9 The dialogue box Calculating Rd(T).

S/No	Secondary steps	Analytical Instructions
		<p>In the new dialogue box that appears (Figure 9) the program enables the accurate calculation of the seismic coefficient. Select the following:</p> <p>Zone: II</p> <p>Importance: I2</p> <p>Soil: S2</p> <p>Tx: 0.15</p> <p>Ty: 0.15</p> <p>Change the value of the B(d0) to 3.5. Finally press [OK] to close the auxiliary window for the seismic coefficient.</p> <p>In the box 'LL Participation during earthquake' type the value 0.30.</p> <p>Your work with the Properties File is now complete.</p> <p>Press [OK] on the bottom right of the dialogue box.</p>
12	<p><b>The file</b> <b>E16S400.YLI</b></p>	<p>In a real project's conditions you should create a specific file in order to specify the soil parameters, the loads combinations of the foundation elements etc. In our example leave the by definition values and move to the next step, which is the specification of the members of the structural system.</p>

## 5<sup>th</sup> STEP: Description of the structural system: Insertion of the elements in the AutoCAD environment.


S/No	Secondary steps	Analytical Instructions
13	<p><b>Move in the starting level.</b></p> <p><b>Rectangular Shaped Column Specification</b></p>	<p>Make sure you are in Level 3 by clicking the index in the AutoCAD command line. If you are not, you can easily move by clicking on the icon  or the  of the STRUCTURE toolbar, if you wish to move upwards or downwards.</p> <p>In order to specify the first rectangular shaped column follow the process:</p> <p style="padding-left: 40px;">Select the <b>MODEL&gt;COLUMN&gt;RECTANGULAR</b> command,</p> <p style="padding-left: 40px;">LC on the icon  of the COLUMNS toolbar.</p> <p>The program corresponds in the toolbar by showing the message {Insertion Point}. Type the value '5.5' and press [Enter].</p> <p>The program shows the message {Angle}. Type the value '0' and press [Enter]. You can see that the first column with 50x50 section is drawn in the plan of Level 3 with the fixed point in the point '5,5' and the sign 'C1'.</p>
14	<p><b>Specification of the second rectangular shaped column</b></p>	<p>Press [Enter] again in order to re-perform the same command (insert rectangular shaped column).</p> <p>In the command line type the value '9,5' and press [Enter] twice. The first for the insertion of the coordinates of the fixed point of the second column and the second for the acceptance of the by definition value of the angle, which is (0).</p>

**15**      **Specification of the third rectangular shaped column**

Three columns constitute the line. Here you can specify the third, by reproducing the second (5) meters on the right.

Perform one of the following:

Select the **MODEL>COPY ENTITIES** command,

LC on the icon  of the AutoSTRAD toolbar.

The program asks you to select the element by showing the message {Select Objects}.

LC on the column 'C2' and [Enter].

If the snaps are not activated LDC on the 'OSNAP' index in the STATUS BAR. In the dialogue box that follows select ENDPOINT and INTERSECTION and press [OK].

Take advantage of this snap and select the fixed point of the column C2.


In the command line type '@5,0' and press [Enter]. You can see that the third column is drawn in the plan, in the right position with the sign C3.

**16**      **Copy of the line in a distance parallel to the first one.**

LC on the **MODEL>COPY ENTITIES** command.

Select the three columns and press [Enter].

S/No	Secondary steps	Analytical Instructions
		<p>In the 'Base point' command LC on the fixed point of C1.</p> <p>Activate the ORTHO by pressing the [F8] key of the keyboard.</p> <p>In the command 'Displacement point' LC at a point in a distance upwards the y-y axes, equal to 5 prosody units. The program reproduces the first line in a distance of five meters towards the positive flow of the y-y axes.</p> <p>See the index of the coordinates in the bottom left part of the AutoCAD window.</p> <p>The prosody units or drawing units of the AutoCAD can correspond in anything. Here you can assume that they correspond in meters (m).</p> <p>Press [Enter].</p>
17	<b>Creation of the third line.</b>	<p>Repeat the same process as above, but now for a distance equal with 9. The program reproduces the first line in a distance of 9 meters lengthwise of the positive part of the y-y axes.</p> <p>See that a grid with 9 rectangular shaped columns. Each column's dimensions are the ones specified at the third step (3<sup>rd</sup> STEP: The preparation for the specification of the structural system).</p>
18	<b>Modify a column</b>	<p>Select the <b>MODEL&gt;COLUMN&gt;MODIFY</b> command. By LC select the C1.</p> <p>Press [Enter].</p> <p>LC on the card 'GEOMETRICAL DATA'</p> <p>Change B's value to 1.5.</p> <p>Press [OK].</p>

S/No	Secondary steps	Analytical Instructions
19	<b>Modify a column</b>	<p>Select the <b>MODEL&gt;COLUMN&gt;CHANGE FIXED POINT</b> command. Change the C3's fixed point by LC on its bottom right corner. Then select the <b>MODEL&gt;COLUMN&gt;MODIFY</b> command from the menu. By LC select the C3.</p> <p>Press [Enter].</p> <p>LC on the card 'GEOMETRICAL DATA'.</p> <p>Change D value to 1.5.</p> <p>Press [OK].</p>
20	<b>Creation of a complex column</b>	<p>Use again the <b>MODEL&gt;COPY ENTITIES</b> command. Select C2. At the 'Base point' command, LC on the fixed point. In the "Displacement point" command, type '@2,0' and [Enter]. A copy of the C2 is created with the serial number C10. Change the B value of the C10 to 1.0 (Model&gt;Column&gt;Modify).</p> <p>With the appropriate use of the command <b>MODIFY&gt;MOVE</b> you can place the new column so that a right angle is created between the C1 and the C10.</p> <p>With the icon  (MODEL&gt;COLUMN&gt;GROUP) from the COLUMNS toolbar and by double LC select the C1 and the new column (C10). Press [Enter]. Those two independent columns group in the complex C1.</p>
21	<b>Specification of a complex Column (Core) from the standardised library.</b>	<p>In the C8's position a core is placed. It would be useful if you could draw two lines (LINE command) that would intersect in the top left corner of the rectangular C8 (that already exists).</p> <p>Select the <b>MODIFY&gt;ERASE</b> command and erase the C8 without erasing the lines.</p>

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

Select the **MODEL>COLUMN>CORE-SHAPED** command.

In the command 'Angle' type '0' and press [ENTER].

With the snap 'Endpoint' chose the intersection point of the lines and press [Enter].

The box of the following figure appears.

From the shape's drawings select the third of the top line (as it is shown in the figure) and press [OK].

See that the core is placed as a complex column with its fixed point in the intersection of the lines.

Select the **MODEL>COLUMN>CHANGE FIXED POINT** command and LC in the top right corner of the C9.

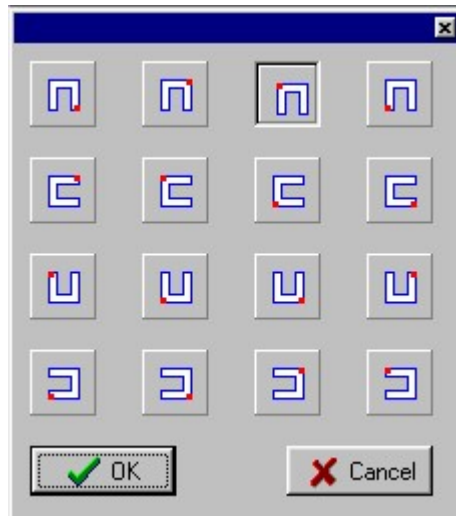


Figure 10 The choice box for the fixed point of the complex column.


S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

Repeat the same process for the creation of the complex column that you did in the C1 and convert the C3 in a complex corner.

When you complete this, the columns should be arranged like the drawing below.

**22 Beams Specification** Perform one of the following:

Select the **MODEL>BEAM>SPECIFYING END POINTS** command.

LC on the icon  of the BEAM toolbar.

The program sees for the first point in the command line. With the 'Endpoint' snap select the right corner of the horizontal part of the C1. As a second point select the bottom left corner of the C2.

For a third point specify a random internal point towards the center of gravity of the plan. The first beam, which is named as B1 by the program, is between C1 and C2.

In the same way specify a second beam, the B2.

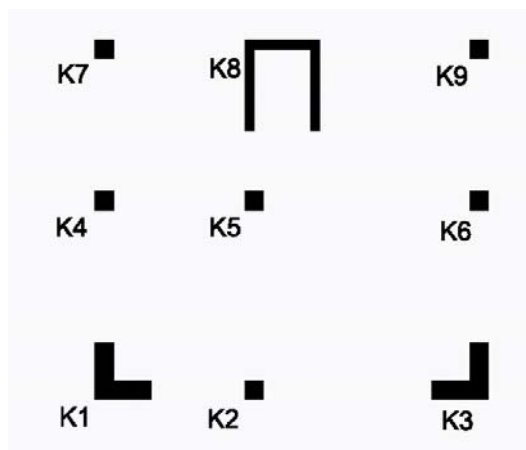



Figure 11 The column's grid at the third level.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

For columns that are in the same straight you can specify a continuous beam by using the command for beam's insertion one time.

To perform this, LC on the icon  (**BEAM BY SPECIFYING END POINTS WITH INTERMEDIATE SUPPORTS**) from the BEAM toolbar.

Select the top right corner of the C3 as your first point and the bottom right corner of the C9 as your second point. LC anywhere on the left of this line. See that the program recognises the column C6, which is in-between, and divides the continuous beam in the spans B3 and B4.

By the combination of the two methods above complete the beam of the level 3, so as to be the same as the drawing below.

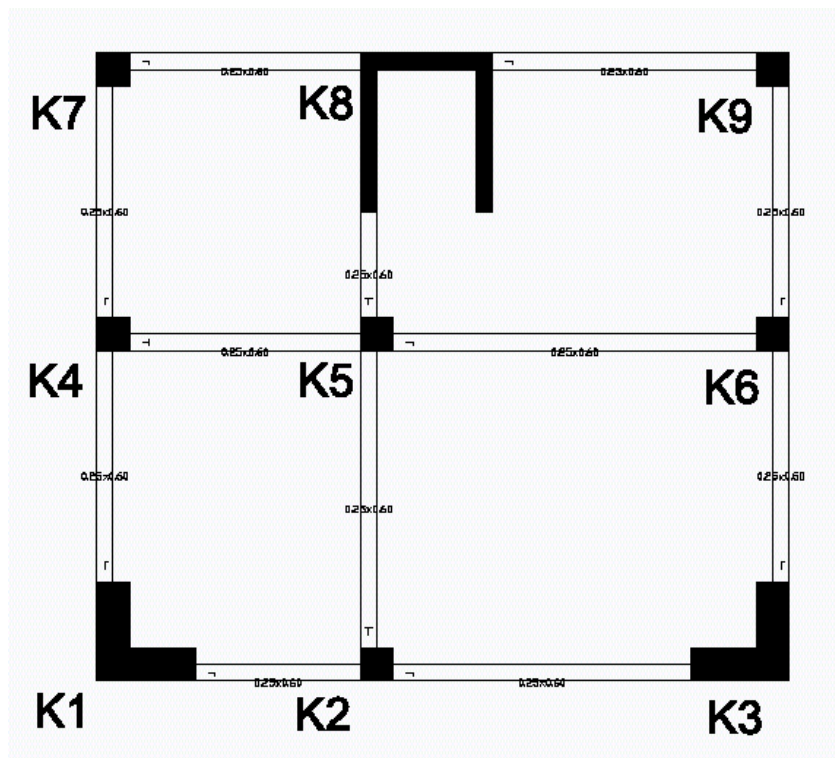


Figure 12 Level 3 after the beam's specification. All the beams have the 25x60 (cm x cm) dimension. The beams in the middle are T-shaped beams, with a load from the infill wall (5000 N/m).

**23 Modify Beam Properties**

Select the **MODEL>BEAM>MODIFY** command and by LC select the middle beams one by one.

Press [Enter].

In the following dialogue box, LC on the card 'GEOMETRICAL DATA' and perform the following modification by marking firstly the corresponding boxes.


Member type: T. Press [OK].

LC on the card 'Beam Loads' and perform the following modification:

Infill Wall Load: 5000.

**24 Copy the specified entities in the level 2 (Basement Roof)**

You can work in the level of the basement roof in the same way as at Level 3. You can also do something better: copy the elements from Level 3 that you wish in Level 2.

LC on the icon  of the STRUCTURE toolbar.

With repeated LCs select all the columns and the middle beams without choosing the surrounding. Press [Enter].

The program creates copies of the chosen elements and places them in the same place in Level 2.

**25 Creation of the Foundation from the level 2 elements**

Press [Enter] again. The command that you have already used activates again. Select with a crossing window (from bottom right up to top left) all the elements of Level 2 and press [Enter].

The dialogue box of the figure below appears.

In the dialogue box COPY TO FOUNDATION LEVEL select all the check Boxes and type (if there is a different value) the value '0.6' in the cantilever length.

See that the program draws footings instead of columns, by applying the value 0.6m for the length of the cantilever of each element, while it diverts the beams to grade beams, I shaped (rectangular).

With **MODEL>BEAM>MODIFY** select all the beams. LC on the beam loads and type 0 in the Infill wall load and press OK.

**MODEL>COLUMN>EXTEND** for choosing the lateral lines of the core's footing and moving them until the core is surrounded by its footing.

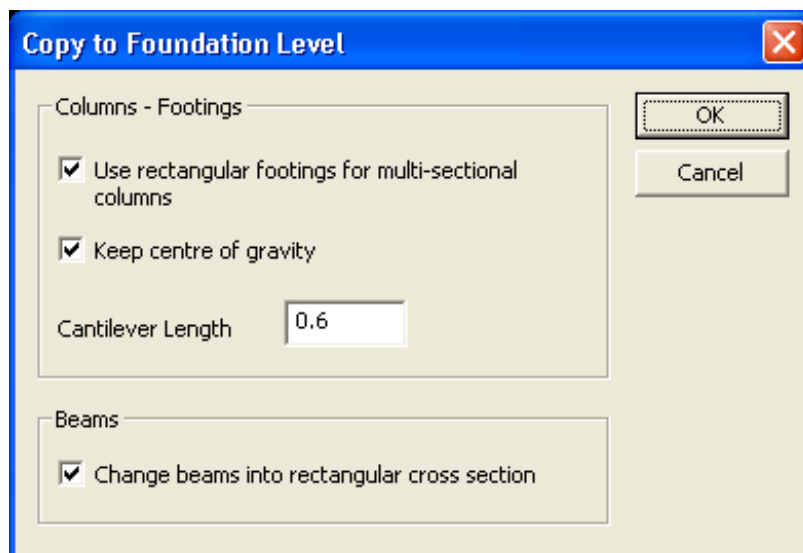








Figure 13 The COPY TO FOUNDATION LEVEL dialogue box.

S/No	Secondary steps	Analytical Instructions
26	<b>Specification of basement shear walls</b>	<p>Move on to the basement roof level by LC on the icon  of the STRUCTURE toolbar.</p> <p>In order to specify the stiffness of the perimetric shear walls, between the perimetric columns, perform one of the following:</p> <p style="padding-left: 40px;">Select the <b>MODEL&gt;BASEMENT SHEAR WALL&gt;SPECIFYING END POINTS</b> command.</p> <p style="padding-left: 40px;">LC on the icon  the AUTOSTRAD MEMBERS toolbar.</p> <p>Just like the specification of the beam by specifying end points, LC on the bottom right corner of the complex column C1.</p> <p>LC on the bottom left corner of the column C2.</p> <p>LC on the top of the semi-plane line. See that the program draws the basement shear walls between C1 and C2.</p> <p>Move on to the specification of the basement shear walls lengthwise the perimeter. Note that it cannot be a basement shear wall with intermediate supports like the beams (you should give all of the them, one by one).</p>

S/No	Secondary steps	Analytical Instructions
27	<b>Formation of the foundation under the basement shear walls</b>	<p>Copy the basement shear walls in the foundation level by transforming them to footings.</p> <p>LC on the icon  of the STRUCTURE toolbar. Select, with repeated LCs the entire specified basement shear walls and press [Enter].</p> <p>In the dialogue box COPY TO THE FOUNDATION LEVEL make sure that all the choices are the same as before. Press [OK].</p> <p>The program draws the foundation (footings) of the shear walls' stiffness by applying cantilever's length 0.6 m.</p>
28	<b>Removal of the stirrup-sections</b>	<p>Select the <b>MODEL&gt;BASEMENT SHEAR WALL&gt;TRIM BASEMENT SHEAR WALL FOUNDATION</b> command.</p> <p>By creating a window select the entire plan of the first level and press [Enter]. The program redraws the footings by removing the sections that get into the other.</p> <p>You just completed the specification of the Space frame elements. In order to complete the specification of the structural system you should specify the slabs at Levels 2 and 3.</p>
29	<b>Refresh the elements of the Space Frame</b>	<p>One of the most important improvements of the Auto-STRAD2001 is the capability for full and accurate supervision of the space frame, as it arises from the specification process of the elements. In cases where the connectivity of the linear elements is altered, you can make corrections with the Refresh commands.</p> <p>Select the <b>MODEL&gt;REFRESH ALL</b> command. The program executes topological checks and makes the appropriate changes.</p>

S/No	Secondary steps	Analytical Instructions
30	<b>3D View of the level's elements</b>	<p>By LC on the icon  you can move into Level 3.</p> <p>From the DRAWINGS toolbar LC on the icon . See in the project window that the structural system appears in 3D.</p> <p>In the toolbar type the command SHADE and press [Enter]. Depending on the adjustments of the structure drawings of your AutoCAD 14 edition, the volumetric drawing of the level 3.</p> <p>LC on the icon  of the DRAWINGS toolbar. The drawing of the space frame elements for third level appears.</p> <p>Search for linear elements that appear with the red color. This is the way that the program uses in order to show the elements that have a connectivity problem.</p>
31	<b>Refresh the elements of the Space Frame</b>	<p>Select the <b>MODEL&gt;REFRESH ALL</b> command. The program corrects their connectivity.</p>
32	<b>Data Checks</b>	<p>When you finish this, select the <b>MODEL&gt;DATA CHECKS</b> command.</p> <p>The program executes a series of checks and notifies in case of a mistake with the corresponding message.</p> <p>In our example there should not be any message of this kind.</p>

S/No	Secondary steps	Analytical Instructions
33	<b>Specification of the slab elements in Level 3 – Cantilever specification</b>	<p>Move on to Level 3 (if you are not already).</p> <p>Select the <b>MODEL&gt;SLABS&gt;CANTILEVERS</b> command.</p> <p>Specify the first point of the cantilever with the snap END-POINT in the bottom right corner of the C1. Specify the second point in a distance - vertical to the beam B1 - (cantilever width) equal to 1.9. Specify the third point in a distance equal to 6 on the right of the second point and parallel to the beam. Specify the last point of the cantilever with the PERPENDICULAR snap in the point of the beam B2 (between the C2 and the C3).</p> <p>Press [Enter].</p>
34	<b>Specification of a hole inside the slab</b>	<p>Select the <b>MODEL&gt;SLABS&gt;HOLES</b> command.</p> <p>Specify the first point of the hole in the internal face of the beam B7 in a 1.05 distance of the C3. Specify the second point in a distance equal to 0.9 towards the center of the slab. Specify the third point in a distance equal to 0.9 from the second point and parallel to the beam. Specify the last point on the same beam.</p> <p>Press [Enter].</p>
35	<b>Specify the analysis strips</b>	<p>Select the <b>MODEL&gt;SLABS&gt;ANALYSIS STRIPS</b> command. The analysis strip is specified as a simple line of AutoCAD 14. Specify the analysis strips so as the surrounding beams, the cantilever and the hole intersects each other (see the figure below).</p>

**36 Slab's and cantilever's recognition**

Select the **MODEL>SLABS>SLABS RECOGNITION** command.

LC on a random point inside the perimeter and in one of the closed rectangular that will be the slabs and the cantilever.

The program recognises the perimeter and numbers the slabs according to their recognition order.

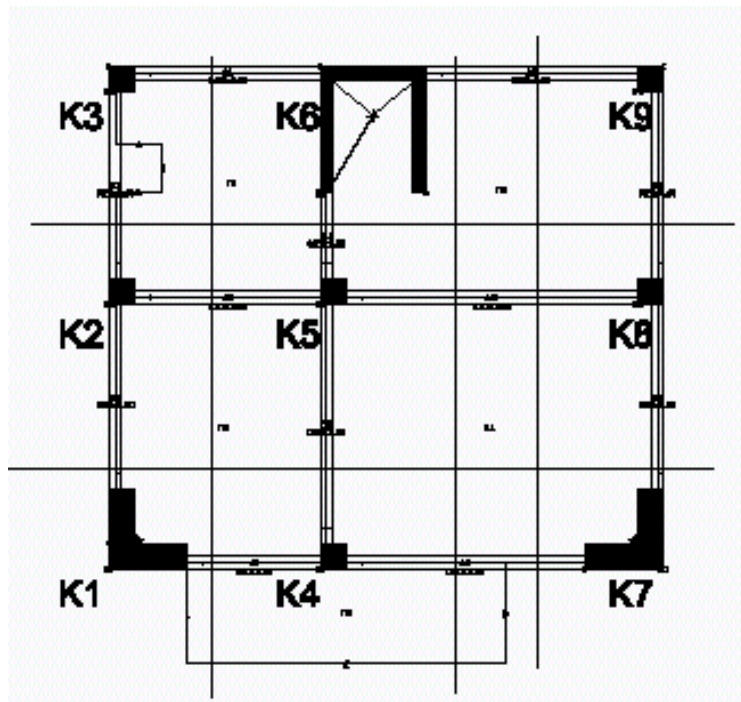


Figure 14 The plan of Level 3 after the slab's specification.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

- 37 Slab's Specification at Level 2** Move on to Level 2 (Basement Floor). This level is different from the previous one because the basement shear walls instead of the beams surround the slabs.
- Specify the analysis strips as earlier with the only difference that they should be two in each direction for each slab, on the right and left of each shear wall's gravity center.
- Recognise the four slabs at Level 2.

## 6<sup>th</sup> STEP: The analysis of the structural system's slabs.

S/No	Secondary steps	Analytical Instructions
		<p>To start with the analysis and the design of the slabs in Level 3 follow the process below:</p> <p>When you finish the process for Level 3 move on - by using the changing-level key - to the level of the basement roof and repeat it in order to complete the slab's analysis of that level too.</p>
<b>38</b>	<b>Analysis of the top Level slabs</b>	<p>If you are not already, move on to Level 3.</p> <p>Select the <b>ANALYSIS-DESIGN&gt;SLABS&gt;SLAB ANALYSIS</b> command.</p> <p>Press [Enter].</p> <p>The program analyses all the slabs of this level with the MARCUS method.</p>

Press the [Valid] key for each one of the analysis strips windows that appear (for 5 strips, 5 windows).

Press [OK] in the dialogue box with the multiplication factors, without changing them.

With these dialogue boxes the length of the sections are specified.

Select the **ANALYSIS-DESIGN>SLABS>EDIT REINFORCEMENT** command.

Select the slabs you wish to reinforce by LC. You can see that the reinforcements calculated from the slab analysis are drawn.

S/No	Secondary steps	Analytical Instructions
		<p>Select the <b>ANALYSIS-DESIGN&gt;SLABS&gt;SLAB SLEDERNESS-DEFORMATION</b> command.</p> <p>Press [OK] at the message {All slabs exempted from deformation check} and at the table that follows.</p> <p>At this point all the slenderness checks are completed, for the ULS and the USS. For the last case the results are displayed in the screen, without a corresponding message appearing on the screen, so that the user can decide to either ignore the insufficiency of the slab according those checks or not.</p>

Select the **ANALYSIS-DESIGN>SLABS>BEAM LOADS** command.

Press [OK].



Press [OK] in the dialogue box with the multiplication factors.

You can see the imperative - on the beams from the slabs - load analytically.

## 7<sup>th</sup> STEP: SPACE FRAME


---

S/No	Secondary steps	Analytical Instructions
39	Space frame	<p>Select the <b>ANALYSIS-DESIGN&gt;SPACE FRAME</b> command. The program asks you to select an entity.</p> <p>By LC select one of the columns (the core) and press [Enter].</p> <p>A window appears after a while. Close the window by choosing <b>FILE&gt;EXIT</b>.</p> <p>The program executes checks during the exit from the Space Frame box also. For this reason, if the project is big (on the contrary with this example) a short delay might occur at this point.</p>

- 40 Optical view of the space frame.** In order to view 3D-view of the space frame follow the process:
- By LC on the icon  of the DRAWINGS toolbar. This command shows the 3D-view of the structural system bodies (not the space frame).
- Select the **MISCELLANEOUS>GRAPHICS>SPACE FRAME** command. In this view you can see the model, as it will be analysed. You can also see the elastic pole (blue vertical line after the analysis) and the mass centers and the elastic torsion (rhomb and circle, accordingly) for each level (which oscillates after the earthquake)
- In order to bring back the plan view (for this Level) LC on the icon  of the DRAWINGS toolbar.

## 8<sup>th</sup> STEP: Predesign checks, Analysis and Checks.


---

S/No	Secondary steps	Analytical Instructions
41	<b>Pre-design checks</b>	<p>The predesign checks of the structural system include a range of checks, from which the most important are the Topology check and the check for the length of the beams. The first finds the possible mistakes in the connectivity between the elements of the linear model, while the second locates the beams with a very small length.</p> <p>Begin with the <b>Topology Check</b>.</p>
42	<b>Topology checks</b>	<p>Perform one of the following,</p> <p>Select the <b>ANALYSIS-DESIGN&gt;PREDESIGN CHECKS&gt;GENERAL PREDESIGN CHECKS</b> command. In the prompting in the command line select randomly a column by LC and press [ENTER].</p> <p>LC on the icon  of the WINSTRAD toolbar.</p> <p>The editor of the Predesign checks of the structural system appears.</p>

From the OPTIONS menu select the TOPOLOGY command.  
For the specific example this check should look like the table below.

½ CHECK FOR SAME START AND END (DATAM)
½ CHECK DATAKM
½ CHECK FOR JOINT WITHOUT CONNECTION
½ CHECK CONTINUOUS BEAMS
½ CHECK FOR CONNECTIONS

Table 1 Results of the Topology.

S/No	Secondary steps	Analytical Instructions
43	<b>Check beam's length</b>	<p>Follow the process:</p> <p>From the OPTIONS menu select the BEAM LENGTHS command.</p> <p>In the dialogue box that appears type the value 0.2m in the top box and the value 10m in the bottom box.</p> <p>Press [Continue]. If there are beams whose length is less than 20 cm and more than 10 m, the program finds them and reports them in the window.</p> <p>In our example something like this should not appear.</p> <p>When you complete your project select the command <b>OPTIONS&gt;EXIT</b> to close the window of the Predesign checks.</p>
44	<b>Structural System Analysis</b>	<p>Perform one of the following"</p> <p>Select the <b>ANALYSIS-DESIGN&gt;ANALYSIS</b> command.</p> <p>LC on the icon  of the WINSTRAD toolbar.</p> <p>The dialogue box of the figure below appears.</p>

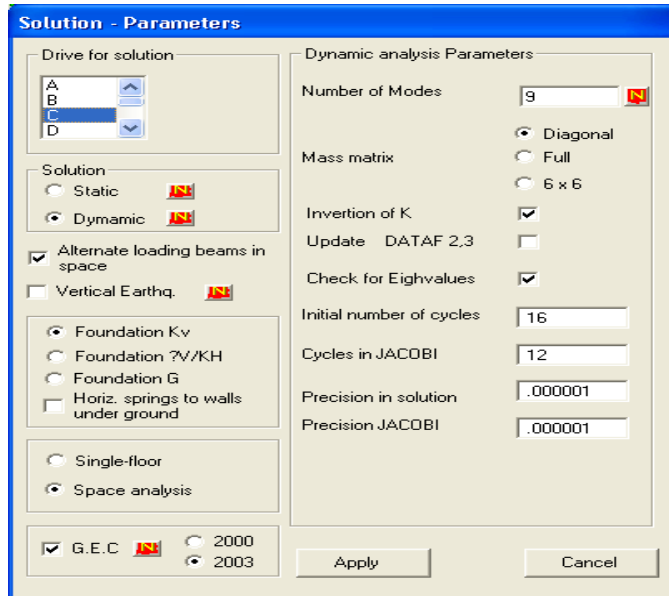


Figure 15 The Solution-Parameters dialogue box.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

In the dialogue box SOLUTION-PARAMETERS perform the following:

Select the hard disc where you installed the program and has a lot of free space, from the list 'drive for solutions'

Make sure that the dynamic analysis is chosen to analyse the project (the program has this information from the default member properties)

Ignore the Accidental Eccentricity and mark the option 'G.A.C 2003'.

On the right of the box type the value '6' as the number of the modes.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

Make sure that the 'Diagonal' choice is marked for the matrix.

Press [Apply].

The program begins to analyse the structural system according to the parameters that you specified and the elements from the default member properties, for each Level.

**45      Solution check**

When the analysis completes a window appears with the results from the Solution check.

We briefly state that the Solution check provides the most accurate information about the progress of the analysis, the modes' response check and the calculated oscillator's eigenperiods.

**46      Structural System Check**

Before you move on to design the members of Reinforced Concrete you should first ensure the contentment of the GAC 2000 criteria. This can be achieved through the Structural System Checks.

Follow the process:

Select the **ANALYSIS-DESIGN>STRUCTURAL SYSTEM CHECKS** command. The window of the figure below appears.

**47      The checks**

To see the results of the torsion sensitivity check (the details of the check with the application of a pair of forces with random eccentricity) press the **[θ]** key.

ACCIDENTAL ECCENTRICITIES										
Lev	Dmax(2)	Dmin(2)	Davg(2)	a(2)	CheckY	Dmax(3)	Dmin(3)	Davg(3)	a(3)	CheckX
1	0,00	0,00	0,00	1,03	OK	0,00	0,00	0,00	1,08	OK
2	-0,56	-0,36	-0,46	1,03	OK	-0,65	-0,39	-0,52	1,08	OK
3	-3,36	-0,40	-1,88	2,21	**	7,12	2,37	4,75	1,56	OK

Figure 16 The Structural System Checks window. You can see the details for the Accidental Eccentricity calculation.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

To see the values of the radius stiffness and the inertia radius and the relevant check select the **OPTIONS>REGULARITY** command.

To see if the criteria for shear wall sufficiency is satisfied select the **OPTIONS>SHEAR WALLS** command.

To see the values of the Center of Gravity and Center of Elasticity components select the **OPTIONS>CG-CE>ANALYSIS** command.

When you finish this, select the **OPTIONS>EXIT** command to close the Structural System Checks window.

## 9<sup>th</sup> STEP: MEMBERS DESIGN.

---


S/No	Secondary steps	Analytical Instructions
48	<b>Specification of the parameters before the member's design</b>	<p>As we mentioned the right order for the members design is:</p> <p>Beam's dimensioning.</p> <p>Column's dimensioning.</p> <p>Foundation member's dimensioning.</p> <p>You must specify the values in some parameters before you start the process of the design.</p> <p>From the Structural System Checks turned up that there are not enough shear walls in the base. You must take it into account during the design of the vertical elements. Therefore you must perform a Check to avoid soft floor and to activate the confinement.</p> <p>Because you are analysing the project with GAC 2000 you must specify the values of the parameters of the foundation elements (footings).</p>
50	<b>Start of the Design Process – Check Table</b>	<p>Perform one of the following:</p> <p>Select the <b>ANALYSIS-DESIGN&gt;DESIGN</b> command and press [Enter].</p> <p>By LC on the icon  of the WinSTRAD toolbar and then by RC. The dialogue box of the figure below appears.</p>



Figure 17 Check table for design.

S/No	Secondary steps	Analytical Instructions
51	<b>Column's Parameters</b>	<p>LC on the [Parameters] key on the right part of the table. The design parameters appear, organised in successive cards.</p> <p>LC on the [Columns] card.</p> <p>In the boxes with the sign 'Check to avoid soft st. mech. form from' type the values '2' to '3'.</p> <p>Activate the 'Confinement' sign.</p>
52	<b>Footings Parameters</b>	<p>LC on the [Footings] card. The footing parameter's card comes upfront.</p> <p>In the section 'Calculation bearing capacity by GAC' select the 'Empirical' and type '0.5' in the box 'Friction coefficient'. Leave the rest without modifications and press [Continue]. The program brings you back to the main design table.</p>

**53      Beams continuity**

LC on the [Cont. Beam] key at the straight that corresponds to 'All'.

The program confirms at this point the continuity of the beams and performs the calculations for the internal forces and moments, at the supports and the spans.

At the shown table for each Level the lines declare the continuous beams, while the numbers in each line's cells declares the serial numbers of the spans (beams) according to the specified order from the 5<sup>th</sup> STEP.

Press [Continue] to move on to the next sheet of the continuous beams until this process is complete.

**54      Beam design**

When you finish with the continuous beams you can begin their design.

Press "As+Beams bars" in the line with the sign "All". With this activity the program will start with the first Beam continuity of the top Level (3) and it will continue until the foundation, where the grade beams will be dimensioned.

The dialogue box for the choice of elements and materials appears. Without any modifications press [Continue].

A big dialogue box appears including the bending moment diagrams of the Continuous beam 1 of Level 3.

At the bottom you can see the reinforcement at the chosen points of the continuous beam (supports, critical areas, spans). To modify the reinforcement, RC on one of those boxes. The small box for the bars' modification appears. Press [Cancel] to close without modifications.

Press [Bending Details] at the top toolbar. The program displays all the actions done for the extract of the bending reinforcement of this continuous beam.

Press [Continue].

The shear force's diagram of the same continuous beam appears.

Right at the bottom you can see the shear reinforcement, in the chosen points of the continuous beam (supports, critical areas, spans).

With [Shear Details] the program shows the calculations that are relevant to the shear.

Press again the [Continue] key. The program moves on to the next continuous beam of this level.

Continue with the same way until you finish with the continuous beams in every level.

## **55 Column's Design**

When you finish with the beam's design you can start the design of the vertical members (columns – shear walls).

Select [Columns] at Level 3. The dialogue box, which allows the choice of a column's group for the design and the possible modification in the Default Member Properties, from where all the parameters are taken, appears.

Without any modifications press [Continue]. The dialogue box for the column's design appears. It must look like the one at the figure below.

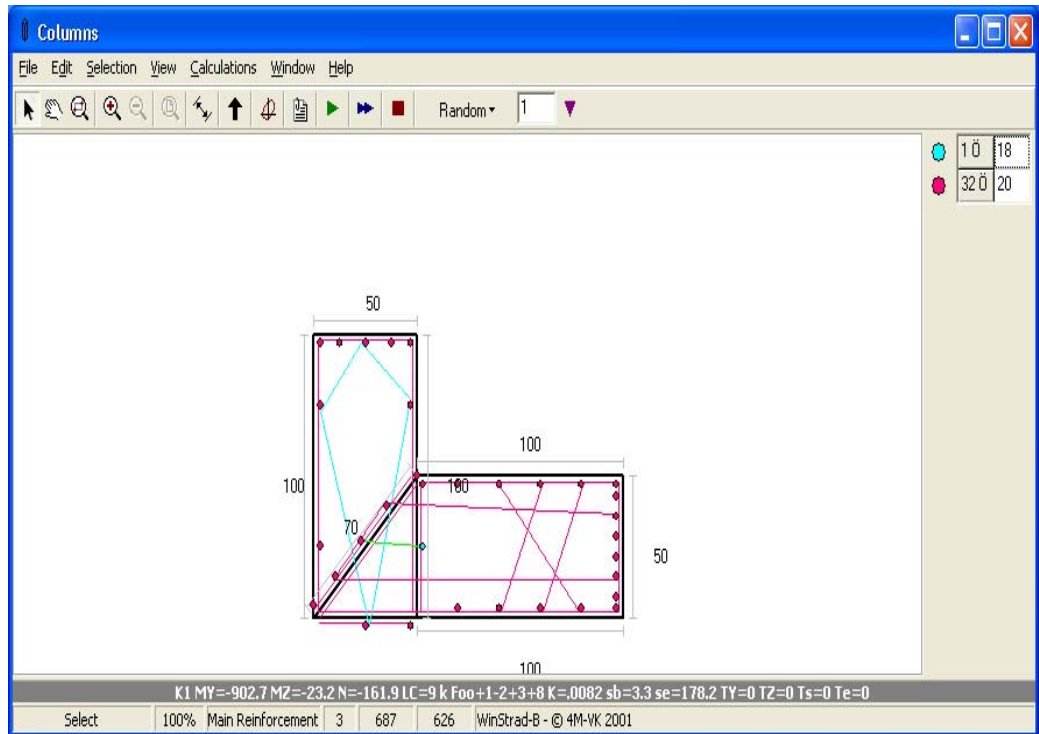


Figure 18 The dialogue box for the column's design.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

The program starts the column's design from the first one according to the specification (C1). The dimensions and the reinforcement appear in the center of the screen; while at the right part are the diameter and the number of bars from the main bar.

The diagonal blue line states the neutral line (which divides the tension zone from the compressive zone).

With the menu commands you can erase some of the given bars, modify the diameter or even add bars at a side or in a random position.

You can see the graphical view of the interaction diagrams of the moments – axial.

With the choice Calculations the program prepares in a text file all the calculations that have been made for the design of this member.

RC on the command 'next stage' (simple arrow). The program moves on to the second stage of this member's design, where the erasure of the stirrups is allowed.

RC again on the 'next member' (double arrow). The program starts the calculations for the next - in the order - column and it shows the results in a similar dialogue box.

Continue with the same way until the design of the columns at Level 3 is complete.

In order to complete the design of the structural system's columns you must repeat the process until the basement roof Level.

**56**      **Foundation Member's Design: Footings**

In order to dimension the foundation elements follow the process:

Press [Footing].

In the following dialogue box press [Continue]. With this activity you state that you wish to start the design of the footings, one by one.

The program corresponds by showing the sketch of C1 with the footing that has been placed. You can see the sign for the reinforcements for every direction and the details about the internal forces and moments (top part of the dialogue box) and the member's dimensions.

Press [Show Stresses] in the right part of the dialogue box that you see. A small list with the signs for the force's combinations appears. LDC on one of them. See that in the sketch there are the developed stresses or R/N, R/V in characteristic points of the footing, which arose from this combination's action.

Press [Continue]. The program shows the next according to the numeration member.



Continue in the same way until you finish with the dimension of the footings.

You can search out for all the results in the PRINTOUT, while the sketch of the entire foundation is created with the **MISCELLANEOUS>DRAWINGS>FORMWORK** command, for Level 1.

**NOTE** In this example for the footings, the member C3 fails because of the Overturning (tension) drift and it has big eccentricity. The member C9 fails because of the drift and the member C10 exceeds the bearing capacity of the ground. For this example press [NO] in the relevant icon for modification the dimensions.

## 10<sup>th</sup> STEP: Results: Issue and Drawings.

---

S/No	Secondary steps	Analytical Instructions
57	<b>Creation of the 3<sup>rd</sup> Level's Formwork</b>	<p>Make sure that you are at Level 3, by checking the command line of the AutoCAD 14.</p> <p>If you are not at Level 3, you can move there easily by clicking on the icon  or the  of the STRUCTURE toolbar, as for the upwards and downwards transition accordingly.</p> <p>In order to create the drawing of the formwork for this level follow the process:</p> <p>Select the <b>MISCELLANEOUS&gt;DRAWINGS&gt;FORMWORK</b> command.</p> <p>The dialogue box of the figure appears.</p>

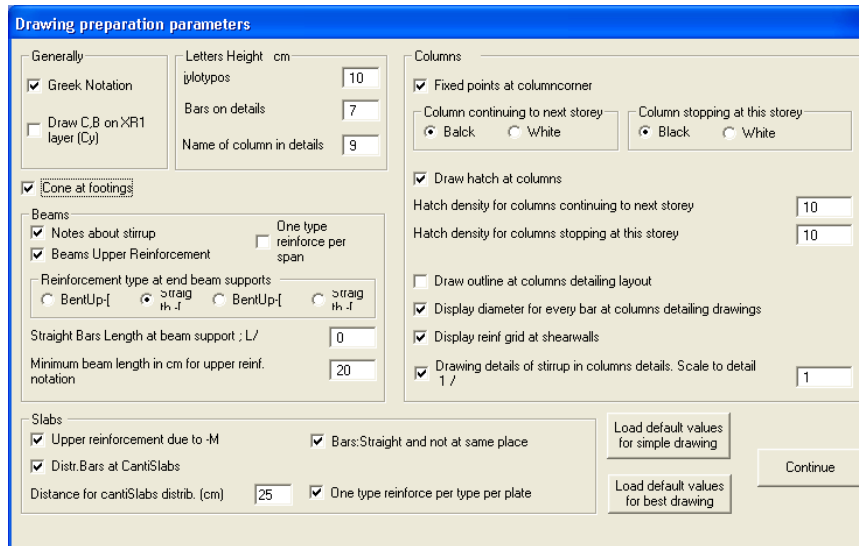


Figure 19 The Drawing Preparation Parameters dialogue box.

S/No	Secondary steps	Analytical Instructions
------	-----------------	-------------------------

58

**Creation of the column's detail drawing at Level 3**

In the Drawing Preparation Parameters dialogue box press [Continue] without any modifications.

The Beam Reinforcement dialogue box appears. Press [Continue] without any modifications.

Observe that the program starts the drawing procedure by taking into account the choices of the previous two dialogue boxes (predetermined) and the dimensioning data that were created for the members of this level, in the previous step.

After a while the name of the DXF file appears on the screen (you can load it by etc.).

You can modify this drawing and print it with a plotter.

In the same way you can easily create the design with the column's details of all columns and the shear walls of Level 3.

Follow the process:

Staying at Level 3, select the **MISCELLANEOUS>DRAWINGS>COLUMN DETAILS** command.

S/No	Secondary steps	Analytical Instructions
		<p>In the Drawing Preparation Parameters dialogue box select the 'Draw outline at columns detailing layout' and the 'Drawing details of stirrup in columns details'. Press [Continue].</p> <p>In the BEAM REINFORCEMENT dialogue box press [Continue].</p> <p>The program creates the drawings for the column details for Level 3 and writes the name of the DXF file. In the question of loading this file replies negatively.</p>
59	<b>Creation of the rest of the drawings</b>	<p>Move on to Level 2 and follow the above process to create the drawing of the formwork and the column's details for this Level.</p> <p>Repeat the same process at the foundation Level.</p> <p>When you finish this you have the complete range of drawings of this project in the catalogue c:\VK\Windows\Stradb\meletes.bld\100.bld\...</p>
60	<b>Technical Report's creation</b>	<p>In order to create a Technical Report, which necessary accompanies the Printouts of your project follow the process:</p> <p>Select the <b>ANALYSIS-DESIGN&gt;TECHNICAL REPORT</b> command. The dialogue box of the figure appears.</p> <p>Complete the elements that you think are necessary and press [OK]. If the software MS Word is installed in the computer it automatically begins and in a few seconds you can see the Technical Report.</p>
S/No	Secondary steps	Analytical Instructions

The text with the Technical Report is ready as a DOC or RTF file if it is created by the WordPad and not by the Word. You can modify it with the same way as any other file of the same type.

**61      Printouts creation**

In order to create a printout follow the process:

Select the **ANALYSIS-DESIGN>PRINTOUT** command.

The driver for collecting the issue's subjects appears. By applying the appropriate choices you can form the text, save it in a RTF or DOC file, modify it and print it.

Select for example [All] under the [Slabs] category. See that the relevant topics will be chosen.

LC on the [Printout] key at the middle of the toolbar.

LC to [Continue] on the small box that appears. With this choice the program gathers all the relevant - for the analysis and design of the slabs of all the structure system's Levels (that have slabs, second and third) - and displays them in a print preview condition on your screen.

You have the capability to modify the text just like you could do with an editor, or if you are not satisfied with these capabilities you can send it at one of the following editors.

<b>S/No</b>	<b>Secondary steps</b>	<b>Analytical Instructions</b>
		<p>The MS Word or the WordPad. The first one should be installed at your computer while the second is installed with the Windows' installation. In order to send the text from the Print Preview to the MS Word (WordPad) LC in the key with the sign of the Word at the left side of the toolbar.</p> <p>In the editor 4M EDITOR that is installed with the STRAD. In order to send the text at the 4M Editor LC on the icon with the red sign [RTF] at the right side of the toolbar.</p> <p>Follow the process to complete the technical report by choosing every time one or more subject categories.</p>