CHAPTER-4

FINITE ELEMENT MODELLING

4.1. GENERAL

The progress of analytical models of the response of RC structures is intricate by the succeeding causes which involve the composite nature of reinforced concrete that comprises of concrete and steel. Concrete shows non-linear behavior even under low level loading due to nonlinear material behavior, cracking and biaxial stiffening.

The finite element method is a beneficial tool for carrying out the analysis and reviewing the behavior of structures. The non-linear fracture analysis exploits the use of mesh generation and the material models in predicting the behavior of materials.

The literature review in the preceding section has suggested that use of a finite element modelling of the RC slabs is undeniably achievable. So it has been decided to apply Atena 3D for the FE modelling. With the aid of this software, study of RC slab has been done.

4.2. PROMINENCE OF FINITE ELEMENT MODELING

The modeling of multifarious behavior of reinforced concrete in its non-linear zone is quite a tedious task. Here comes the application of finite element method that helps in investigating the initiation of cracks, the development of cracks and the post-cracking behavior of any structure or the structure member. The method can be used to study the behavior of reinforced and pre-stressed concrete structures including both force and stress redistribution. It is a valuable tool for locating the load deflection behavior and its crack forms in numerous loading situations.

Therefore, the finite element method has become a powerful computational tool, which allows complex studies of the nonlinear response of RC structures to be carried out in a routine fashion. With this method the importance and interaction of different nonlinear effects on the response of RC structures can be studied analytically.

The software simulates the behavior of concrete and reinforced concrete. It offers a vast range of modeling in 2D and 3D material models.

The software has three main functions:

1. Pre-processing. This step involves the input of geometrical objects (concrete, reinforcement, etc.), loading conditions as well as the boundary conditions, meshing and solution parameters.

2. Analysis. It makes possible a real time monitoring of results through calculations.

3. Post-processing. This step comprises an access to an extensive range of graphical and numerical outcomes.

Atena 3D user interface comprises of three key windows, through which a user can control three stages of execution: Pre-processing, Run (calculation) and Post-processing. ATENA has a modular structure. Calculation phases are processed by these program modules separately by opening a corresponding window.

The top menu bar consists of tools like File, Edit, Input, and so on through which additions and changes can be done. The program processing phase on the top right hand side involves the steps of run and post-processing. The access to data comprises of changes and alterations to existing data. The graphical window is the phases in which the model is displayed and the information text bar is at the bottom part of the window where the related information is written.

4.3. MODELING METHODOLOGY

The slab models were generated using Atena 3D software. The outcomes acquired from the analysis of this FE models have been exploited for the comparisons and interpretations.

4.4. TERMINOLOGY IN ATENA 3D

The terms related to the software and that are frequently employed are defined as under:

- Material – Constitutive model of material used in Atena to represent a certain material type.

- Geometrical model – Solid structure described as an assembly of basic geometrical objects (macroelements, their contacts, reinforcing, etc.)
- Macroelement – basic geometrical 3D object defined by surfaces.
- Surface – planar face of object
- Line – line connecting two joints.
- Joint – basic geometrical entity.
- Reinforcement bar – multi-linear object embedded in macroelements.
- Contact – contact plane of two macroelements.
- Load case - load action defined on assembly of macroelements.
- Load step – increment of load action.
- Analysis steps – loading history defined as a sequence of load increments.
- Finite element – basic element used for approximation of deformation and stress state of a solid.
- Node – point in finite element mesh.

4.5. STEPS IN FINITE ELEMENT MODELING

4.5.1. Coordinates:

The software comprises the coordinates on global level in the x, y, and z directions that were set by default.

4.5.2. Material from Direct Definition

Adding new material is done by pressing the button and selecting the direct definition method of material input. In direct definition material type can be selected from a list and its parameters are generated on selection.

Material Properties from Direct Definition

- **Concrete**

In the present investigation, concrete material is modeled as a 3D nonlinear cementitious2 as it exhibits nonlinear behavior.. The material is assigned to the slab using material model "Nonlinear Cementitious 2" (NLC2) with the adjusted material parameters under the direct definition. The physical properties of 3D nonlinear cementitious2 material are given in Table 4.1.

Table 4.1. Material Properties of Concrete

Properties	Values
Elastic Modulus (MPa)	21981.0
Poisson Ratio	0.2
Tensile Strength (MPa)	3.12
Compressive Strength (MPa)	32.0

- **Steel**

For steel, the material model "Bilinear Reinforcement" was assigned under direct definition. HYSD steel of grade Fe-415 of 8 mm diameter are used as main reinforcement and 8 mm bars are also used as distribution reinforcement. The properties of these bars are shown in Table 4.2.

Table 4.2. Material Properties of Reinforcement

Properties	Values
Elastic Modulus (MPa)	200000
Yield Strength (MPa)	415
Specific material weight (kN / m_3)	78.5

4.5.3. Modeling the slab

The slab was made by assigning the coordinates to the joints. The joints were then joined by the lines. The surface was generated by assigning the surface to all the lines of the slab.

4.5.4. Modeling the reinforcement

The reinforcement bars were modeled under the "reinforcement" by adding the joints of reinforcing bar and then joining it by the line segments. Then the reinforcing bar is assigned the property to the material model "Reinforcement". The values given to the diameter and the number of bars change the usage of the material "Reinforcement" from 0 to 1.

4.5.5. Macroelements

Macroelements define geometry of solid objects. The input can be retrieved from the tree that has two sections for data manipulation:

- Individual - one object can be defined, edited or removed. This function serves to input, editing and removing of a macroelement.
- Selected - a group of selected objects can be manipulated. Function of remove, copy and move one or more macroelements can be performed.

4.5.6. Load Cases

The menu of loading provides an access to loading. Loading is imposed on geometry. Following types of load cases are possible:

1. Body forces.
2. Forces.
3. Supports.
4. Prescribed deformations.
5. Temperature.
6. Shrinkage.

7. Pre-stressing.

The prescribed deformation from experimental investigation was considered and given as an input parameter in prescribed deformation.

4.5.7. Run Analysis

Input of solution parameters can be accessed through the data tree shown in Fig. 4.1.

Fig.4.1. Data tree for Run

Solution Parameters

On opening the solution parameters, the window as shown in Fig.4.2. opens on which the data can be edited.

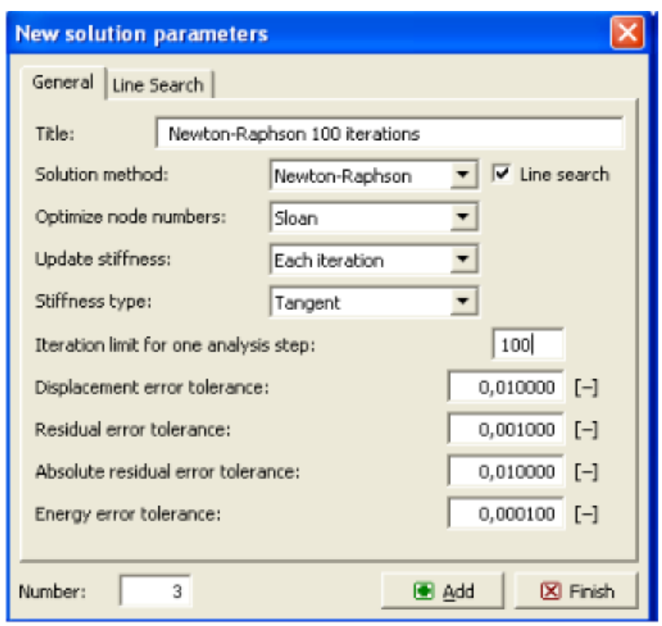

Fig.4.2: Parameters for Solutions

Comparison of the load-displacement diagrams measured in tests and achieved in the numerical simulation has been done. Standard Newton-Raphson method has been carried out.

Analysis steps

Analysis steps define the loading history for solution. Following rules apply to load steps:

1) Load steps are incremental. This means that values of loadings applied in the current step are added to the loading applied in previous load steps.

2) Loading need not be proportional.

3) Loading history is unique.

4) Superposition of stress state is not admissible in nonlinear analysis.

In this dialog following data for load step are entered:

1) Load cases are listed in the first field. Load cases are defined by numbers, separated by comma. Continuous array of load cases can be define by first and last number separated by dash. For example load cases 1,2,3,4 can be defined as 1-4.

2) Construction cases appear in the window “Construction case”. It can be selected from the list, which gets unfolds after clicking.

3) Solution parameters are selected in menu “Sol. Params”. It can be selected from the list, which unfolds after clicking.

4) Analysis step multiplier is applied to the load values defined in load cases.

5) Number of load cases to be generated is defined in “Number of added load steps”.

6) Load steps can be added; parameters can be changed, deleted and inserted between previously defined steps.

Monitoring Points

Monitoring points function to observe results of calculation during analysis. Monitoring points have similar meaning as measuring gauges in laboratory experiments.

4.5.8. Run

The main function of Run window is to control calculation progress (start, stop or suspend analysis) and to monitor intermediate results.

4.5.9. Post-Processing

ATENA post-processing can be started from other two modules (pre-processor, run) by clicking on the pre-processor icon located in the top right tool bar. Postprocessor is also automatically initialized, when load step-data with results are read from the file option of menu item.

4.6. FINITE ELEMENT MODELLING OF SLABS

The modeling of the slabs using ATENA software is shown in Figs. 4.3-4.17.

Fig. 4.3-4.7 show the modeling of control slab S1 in reinforced form and solid model. In Fig.4.3, the slab model has been shown that is formed by giving the coordinates in all the three directions. These coordinates are then joined by lines to generate this model.

Thereafter in Fig. 4.4, the reinforcement is added by giving the coordinates to the line segment. The line segment when give its diameter changes its usage from 0 to 1.

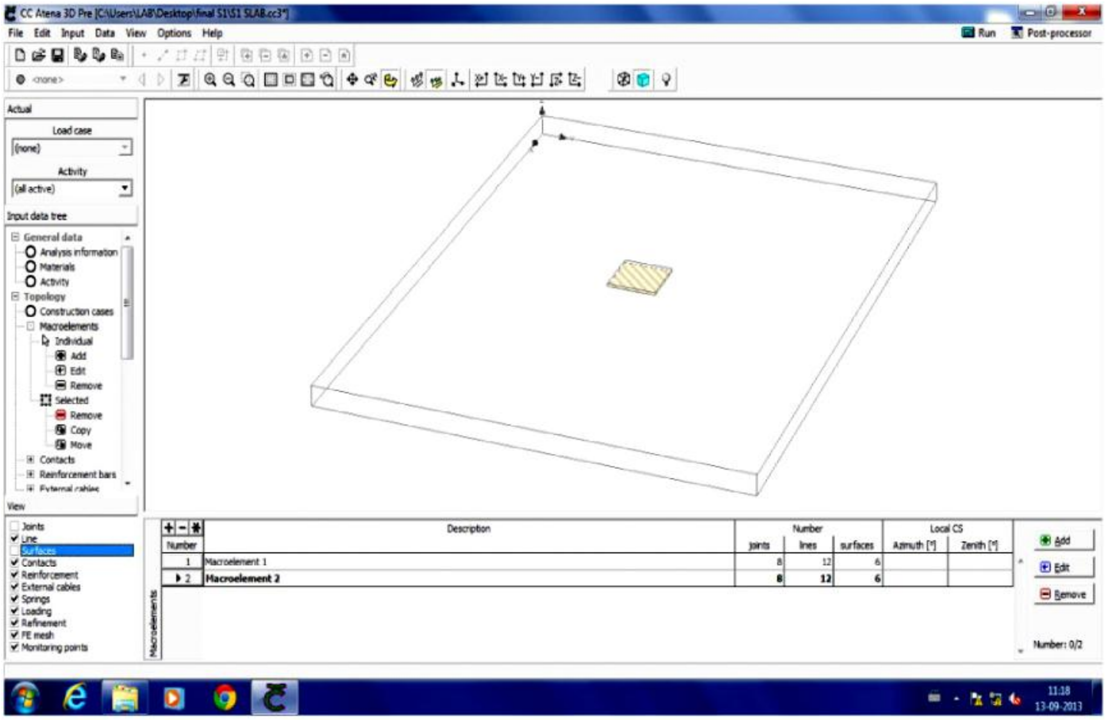

Fig.4.3: Model of slab S1

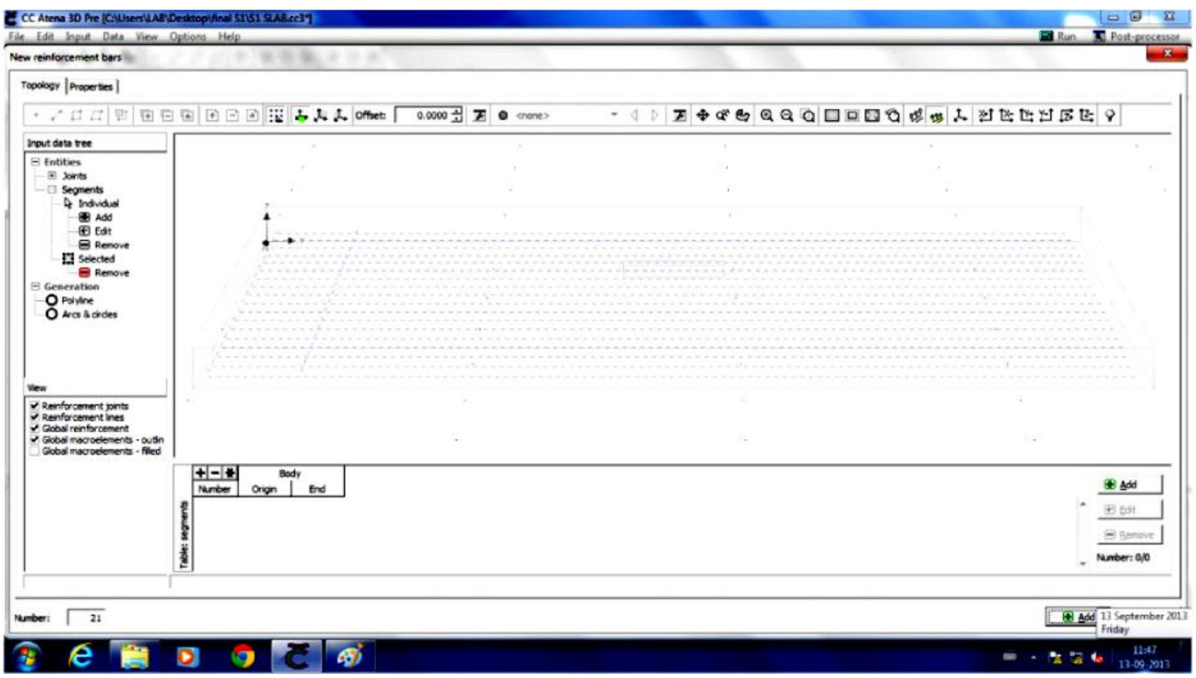

Fig.4.4: Reinforcement in slab S1

Fig.4.5 below indicates the meshed view of the reinforcement in longitudinal and transverse direction.

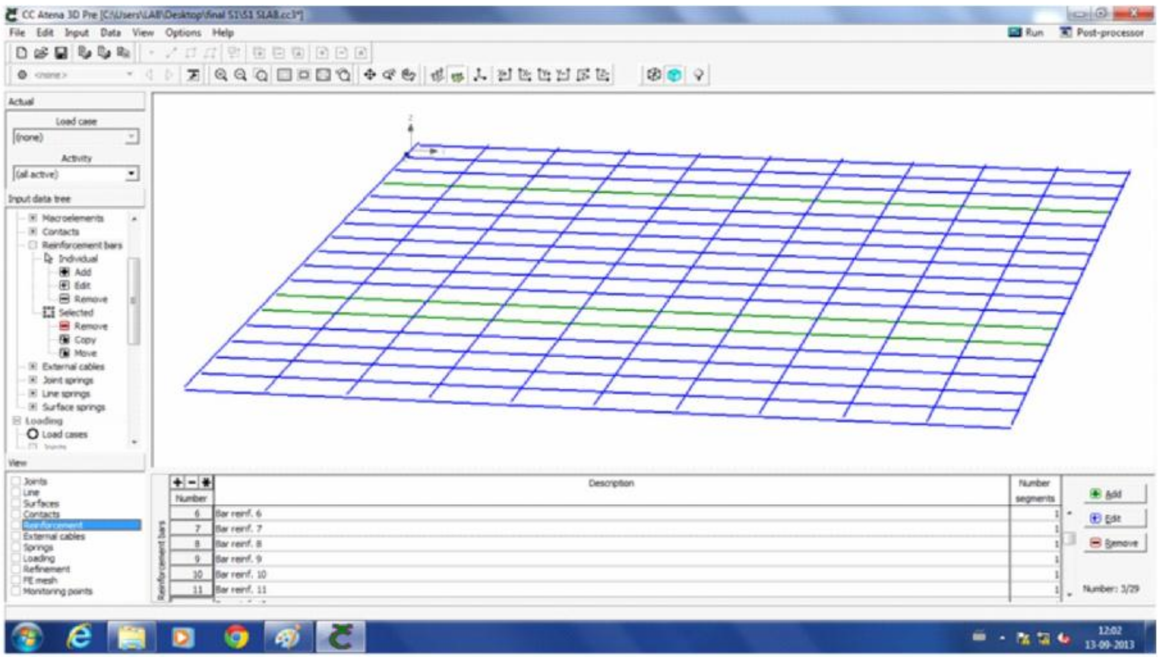

Fig.4.5: Meshed form of Reinforcement in slab S1

Then the solid model is shown in Fig. 4.6. where the surface property is given to the slab. The uniformly distribute load is assigned to the slab by creating a small patch of area around the central point of the slab. So, the complete meshed and solid model of the slab S1 is shown in Fig. 4.6 and 4.7.

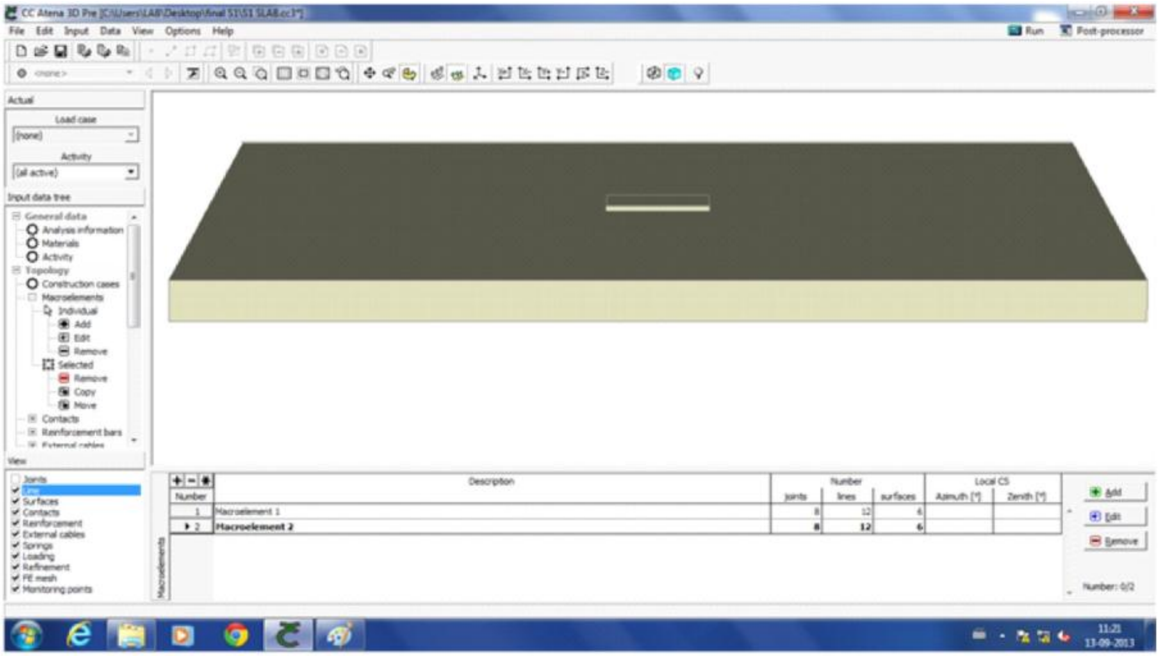

Fig.4.6: Solid Model of slab S1

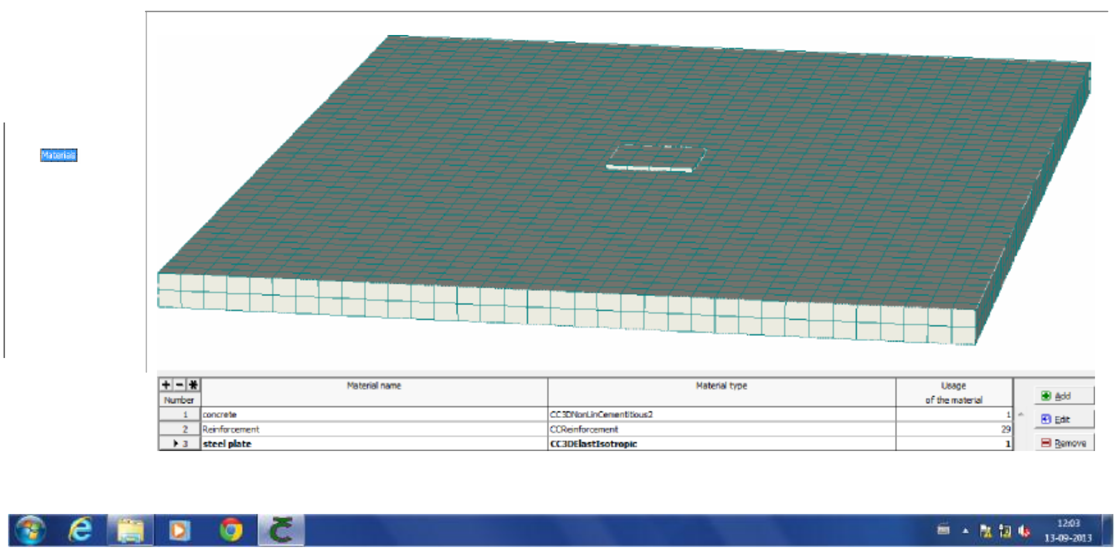

Fig.4.7: Meshed model of Slab S1

Then the modeling of the Slab S1 was done by attaching the materials with specified thickness in the cases of Glass Fibre Reinforced Polymer (GFRP), Ferrocement and Mild Steel plate on the tensile face of the slab. This is better seen in Fig. 4.8-4.11.

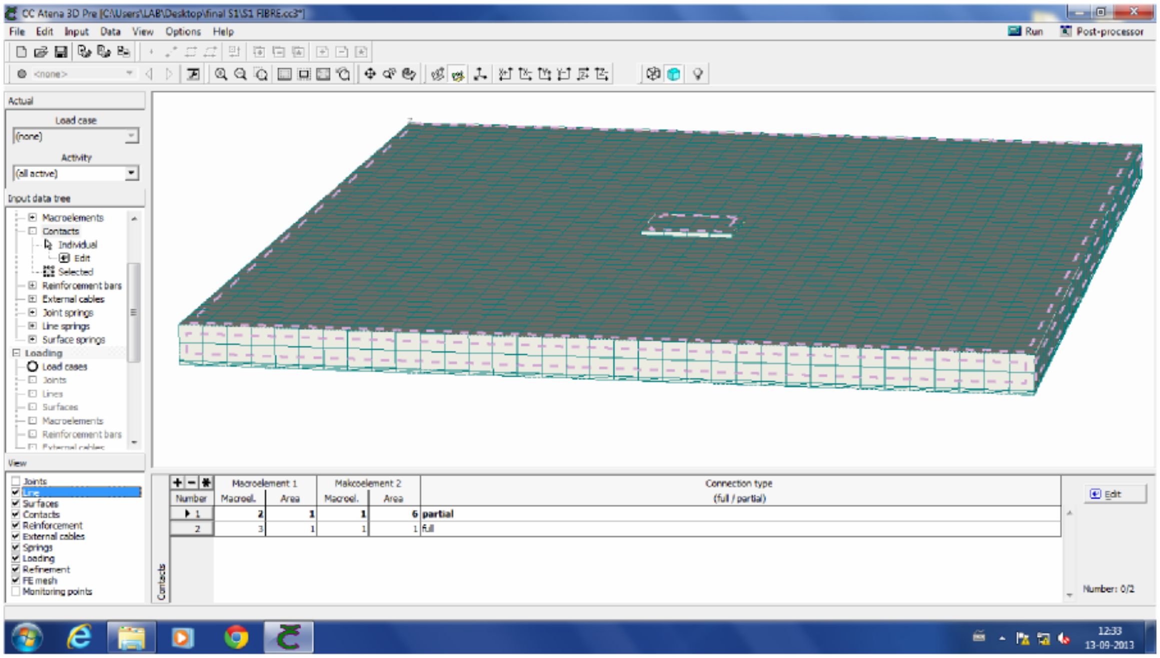

Fig.4.8: Meshed Model of GFRP retrofitted Slab S11

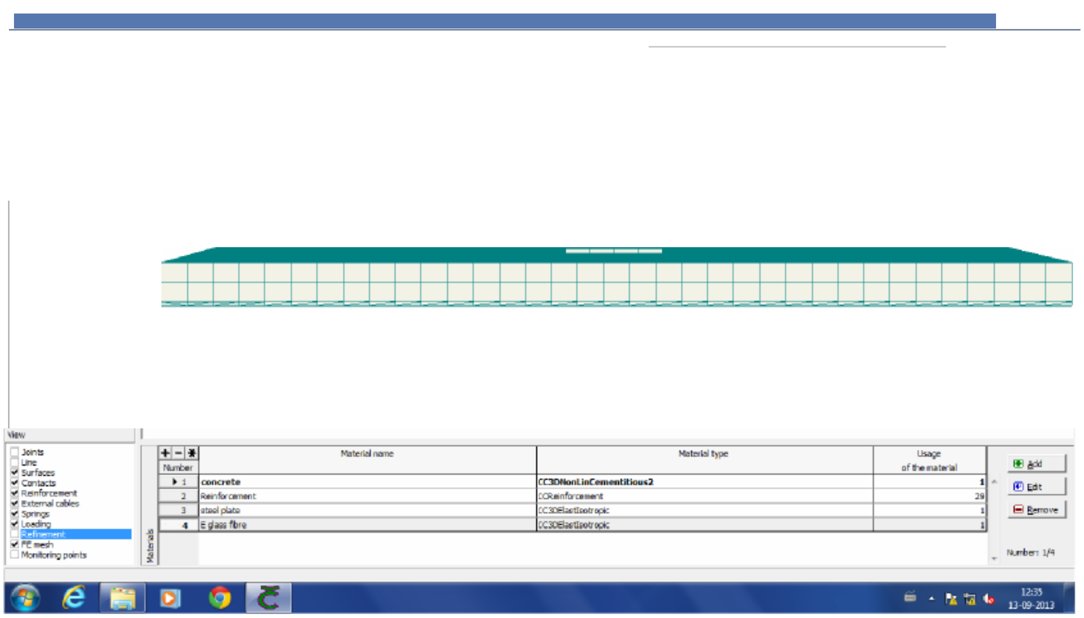

Fig.4.9: Meshed Model of GFRP retrofitted Slab S11

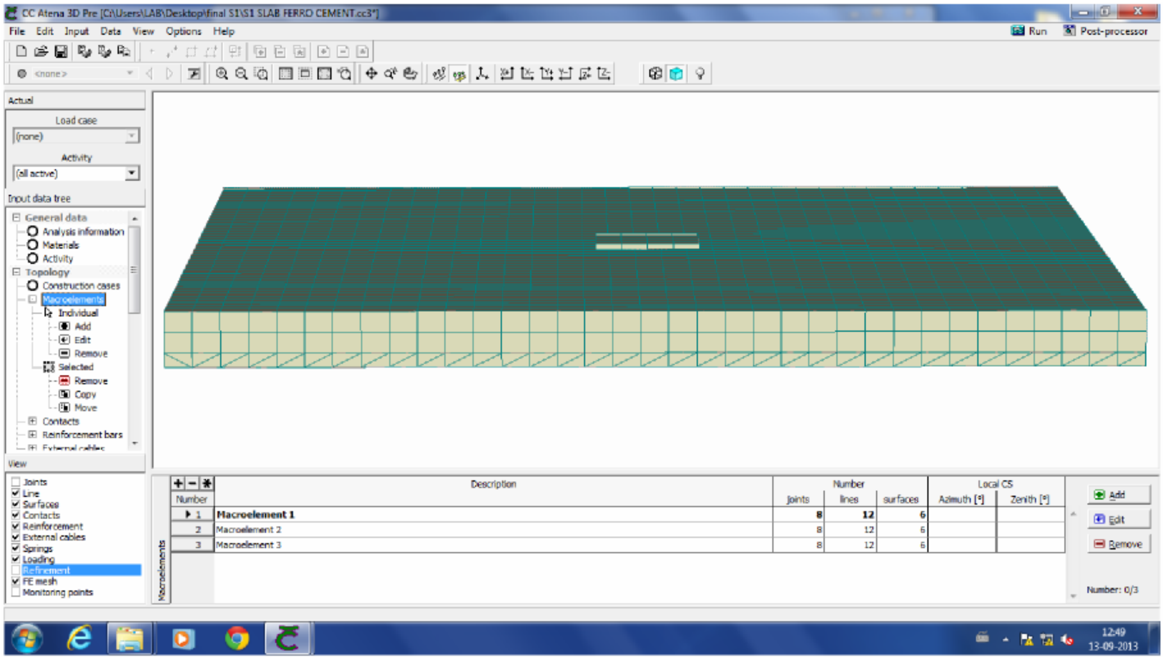

Fig.4.10: Meshed Model of Ferrocement retrofitted Slab S12

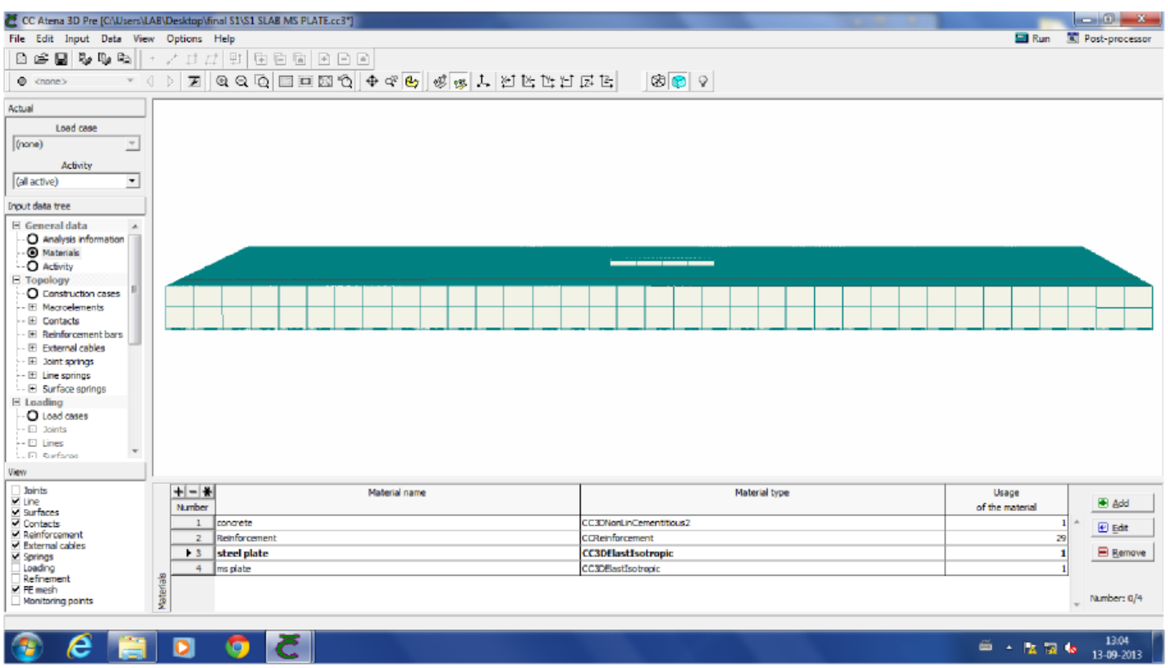

Fig.4.11 Meshed Model of MS plate retrofitted Slab S13

Similarly, the model of Slab S2 in solid and meshed form is seen in Fig.4.12-4.14.

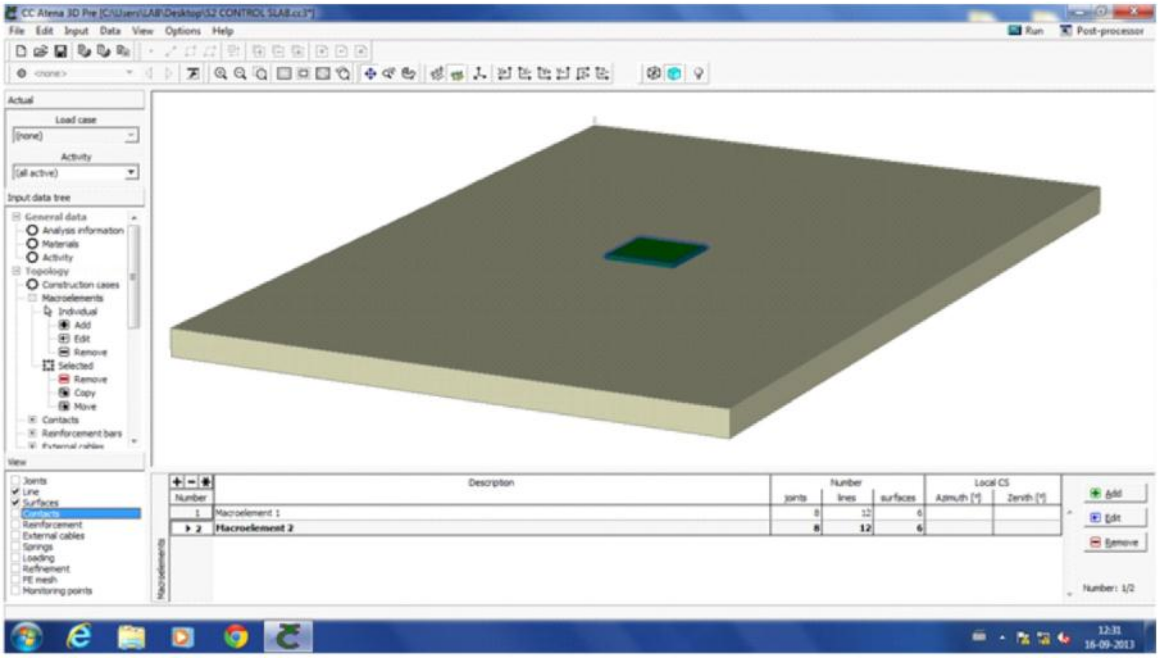

Fig.4.12: Solid Model of slab S2

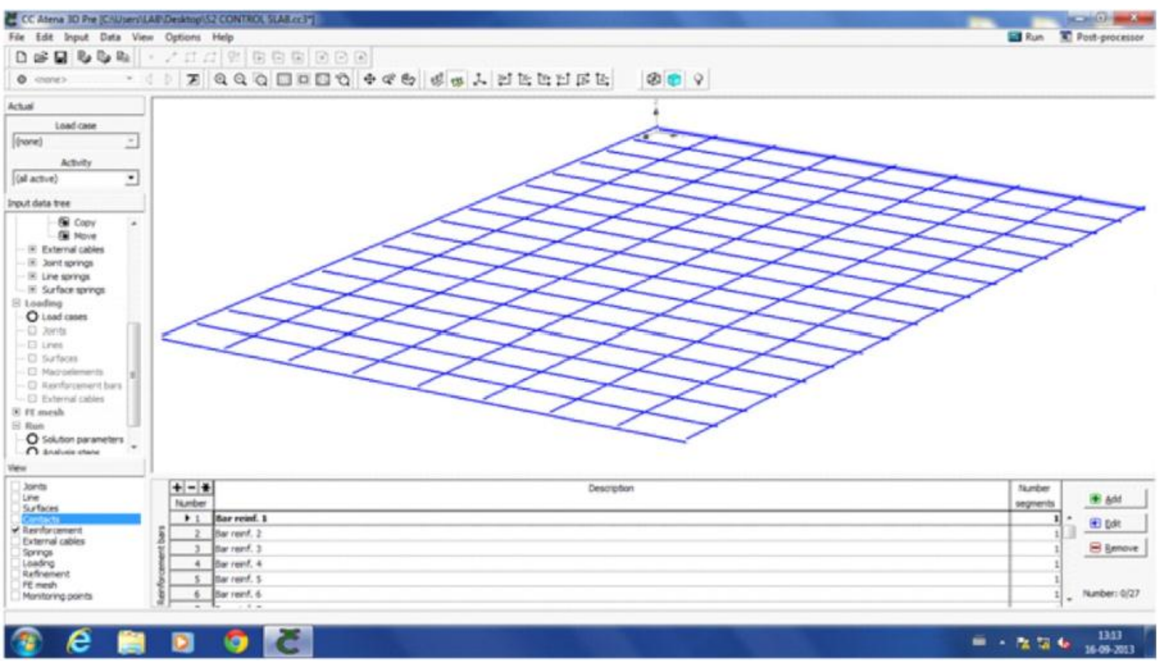

Fig.4.13: Reinforced Model of Slab S2

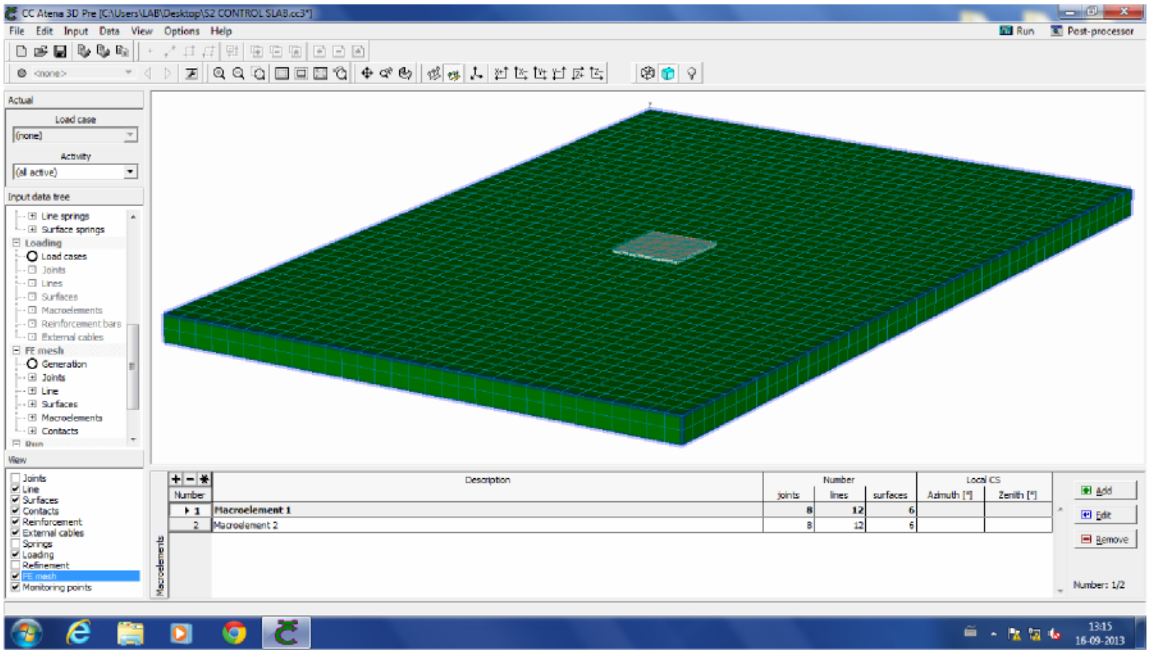

Fig.4.14: Meshed model of Slab S2

The modeling of the slab S2 when retrofitted with GFRP, Ferrocement and Mild Steel Plate is shown in Fig. 4.15- 4.17.

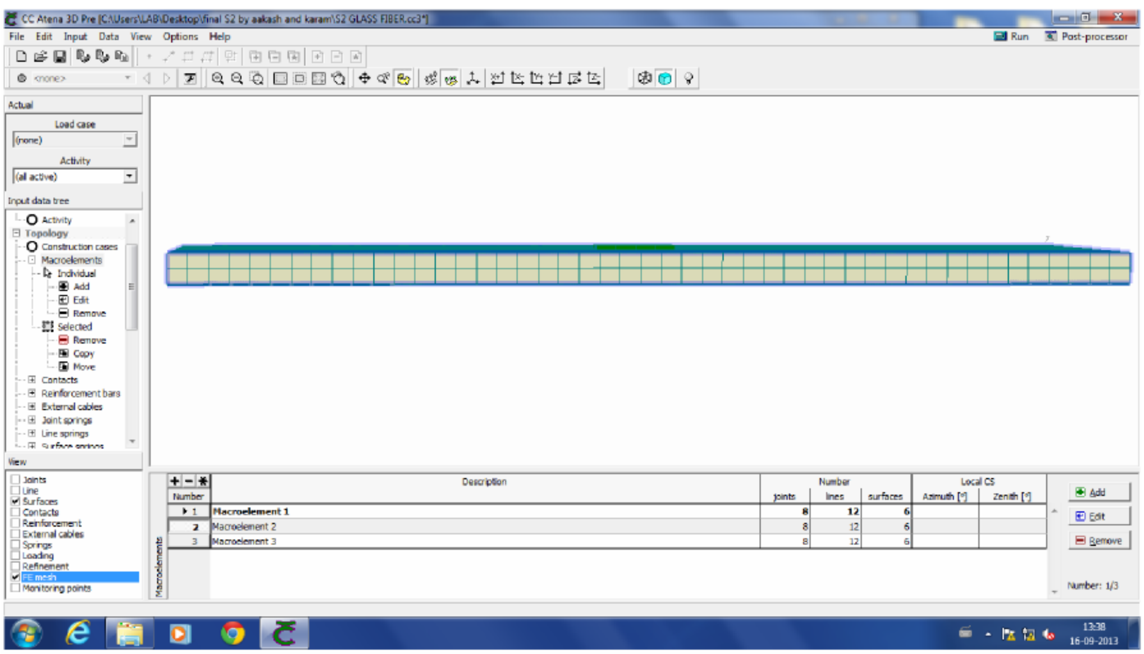

Fig.4.15: Meshed Model of GFRP retrofitted Slab S21

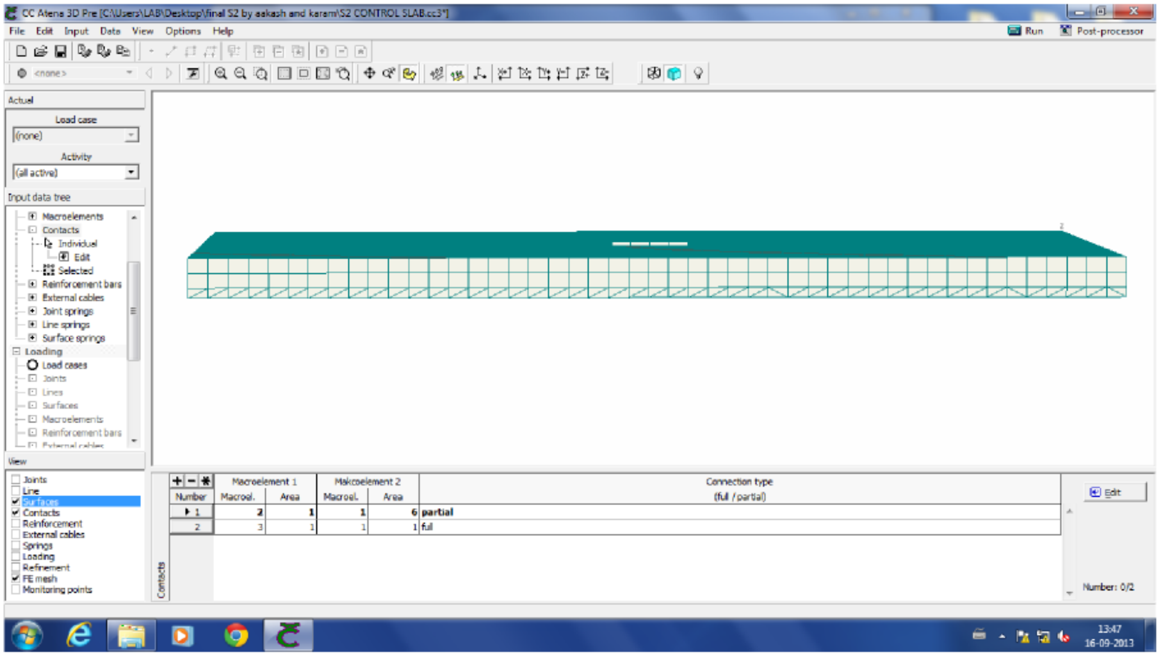

Fig.4.16: Meshed Model of Ferrocement retrofitted Slab S22

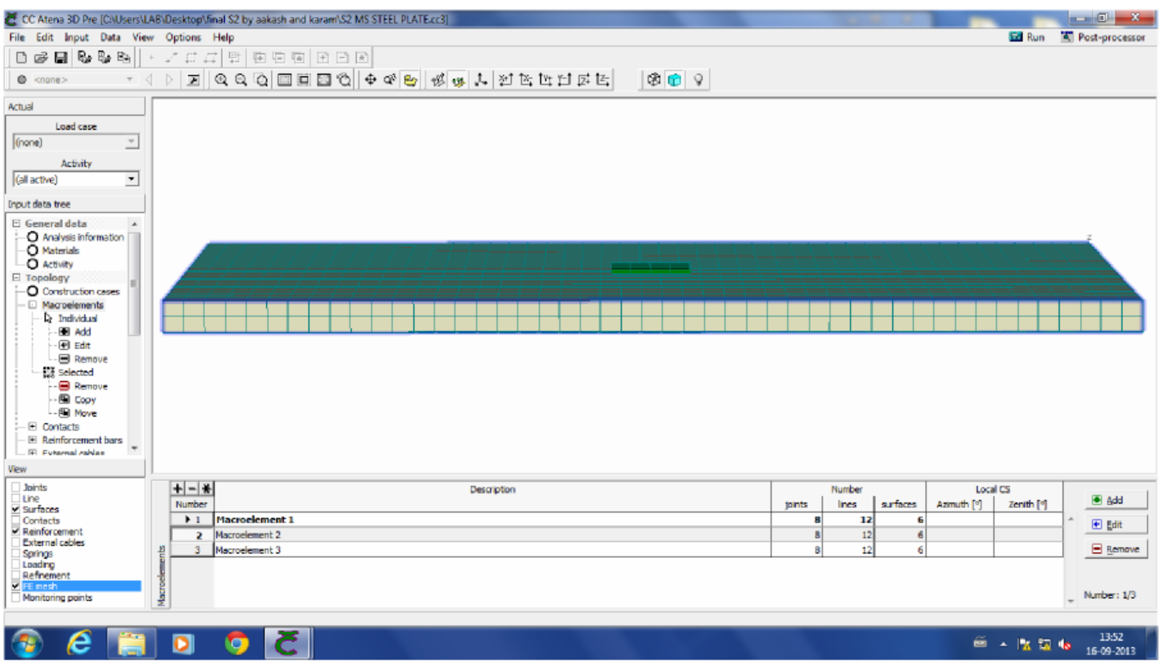

Fig.4.17: Meshed Model of MS plate retrofitted Slab S23.

4.7. CLOSURE

Nonlinear finite element analysis is a resourceful tool for analysis of the reinforced structures. The finite-element analysis was carried out to simulate the nonlinear behavior and failure mechanism for two-way reinforced concrete rectangular slabs. Testing and numerical modeling of control slabs as well as the retrofitted slabs was performed in order to check their performance.

An initial analysis was done on controlled slabs of two different sizes S1 and S2. The Newton- Raphson method was used for nonlinear solutions. Iteration was continued until the convergence for residual displacement caused by unbalanced force was satisfied.

Thereafter, the analysis was done on the retrofitted slabs that were outwardly strengthened with FRP sheets of glass fibres, Ferro cement and MS sheet.

Subsequently, the results of the two analyses were compared and the interpretations made.