# Structural Design with Softwares

**Prepared by**
**Dr. Jafar Sadak Ali**
**(Only for Educational Purpose)**

# CLASSICAL METHODS OF STRUCTURAL ANALYSIS

- Moments Distribution Method
- Slope Deflection Method
- Matrix Method
- FEM Approach

# BASIC SOFTWARES for ANALYSIS

- STAAD Pro (A FINITE ELEMENT PACKAGE)
- ETABS (A FINITE ELEMENT PACKAGE)
- SAP (A FINITE ELEMENT PACKAGE)
- AUTO CAD for drawing purpose

# STAAD PRO

- STAAD Pro is a commercial software used for Analysis and Design of Structures. STAAD Pro was developed for practicing engineers
- Generally CONCRETE Structure & STEEL Structures are analyzed and designed with STAAD Pro.
- It is good and user friendly for linear structures.
- 2D/3D Static Analysis
- Dynamic/Seismic Analysis
- Secondary Analysis

# ETABS

- Linear Static Analysis
- Modal Analysis
- Eigenvector Analysis
- Ritz-Vector Analysis
- Response Spectrum Analysis
- Time History Analysis
- Nonlinear Time History
- Initial P-Delta Analysis
- Nonlinear Static Analysis

# SAP

SAP is a general purpose finite element analysis program for structural analysis. The program can analyze structures that are subject to either static or dynamic loads. The structures can be described in terms of truss, frame, plate, shell, or brick elements or combinations of elements.

- NON LINEARITY (EITHER MATERIAL OR GEOMETRIC)
- DYNAMIC ANLYSIS FOR EARTHQUAKE LOAD
- P-DELTA ANALYSIS,PUSH OVER ANALYSIS

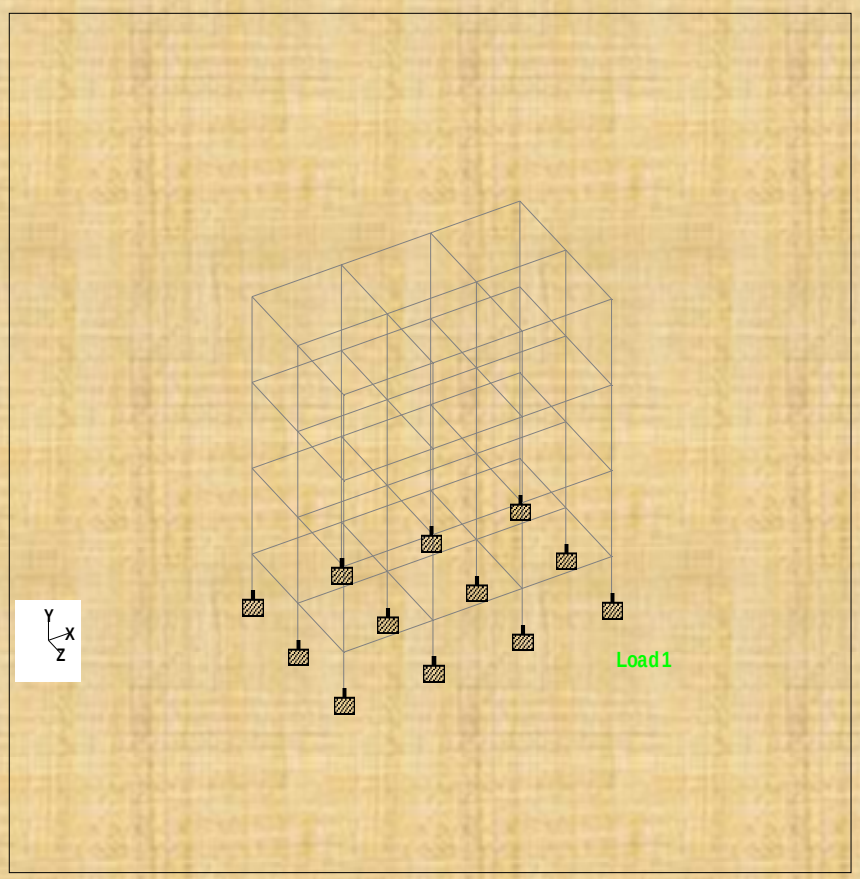
# Steps for design

- Basic Geometry Generation
- Basic/Primary Load Case Generation As per IS:875 (part-II,III,IV) IS:1893(Part-I,IV),IS:800(2007/1984)
- Load Combination Generation as per IS:875(Part-V)(1987)
- Analysis of Structure as per Matrix Method of Analysis with Finite Element Method.
- Design of structure with proper Design Command

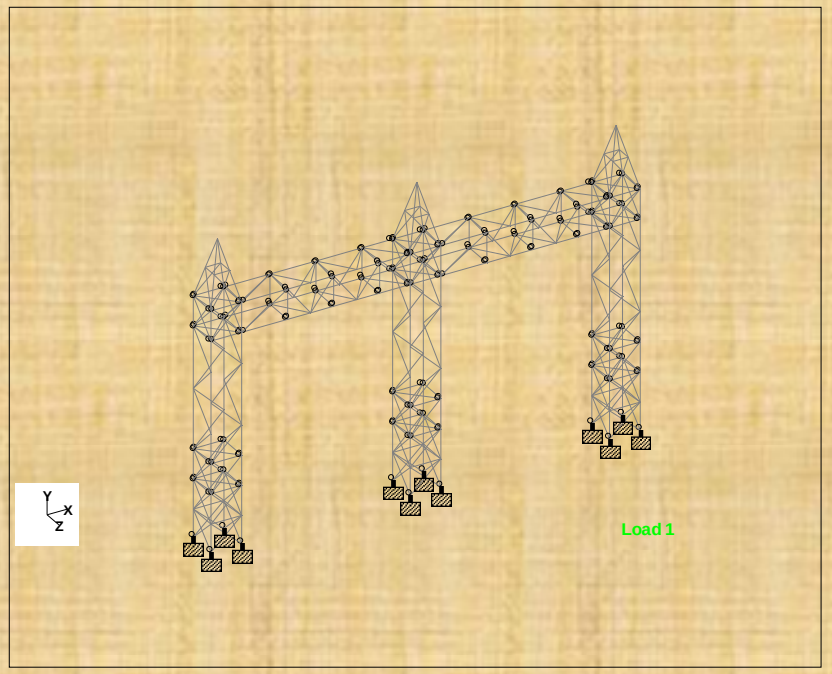
# Basic Geometry Generation

- NODE GENERATION
- NODAL CONNECTIVITY GENERATION
- BEAM/TRUSS/MEMBER SPECIFICATION
- ELASTIC ROPERTIES, MATERIAL CONSTANTS
- PLATE ASSIGNMENT (IF ANY)
- PROPER SUPPORT (BOUNDARY CONDITION) ASSIGNMENT
- ELEMENTAL STIFFNESS MATRIX
- GLOBAL STIFFNESS MATRIX

# STAAD Pro

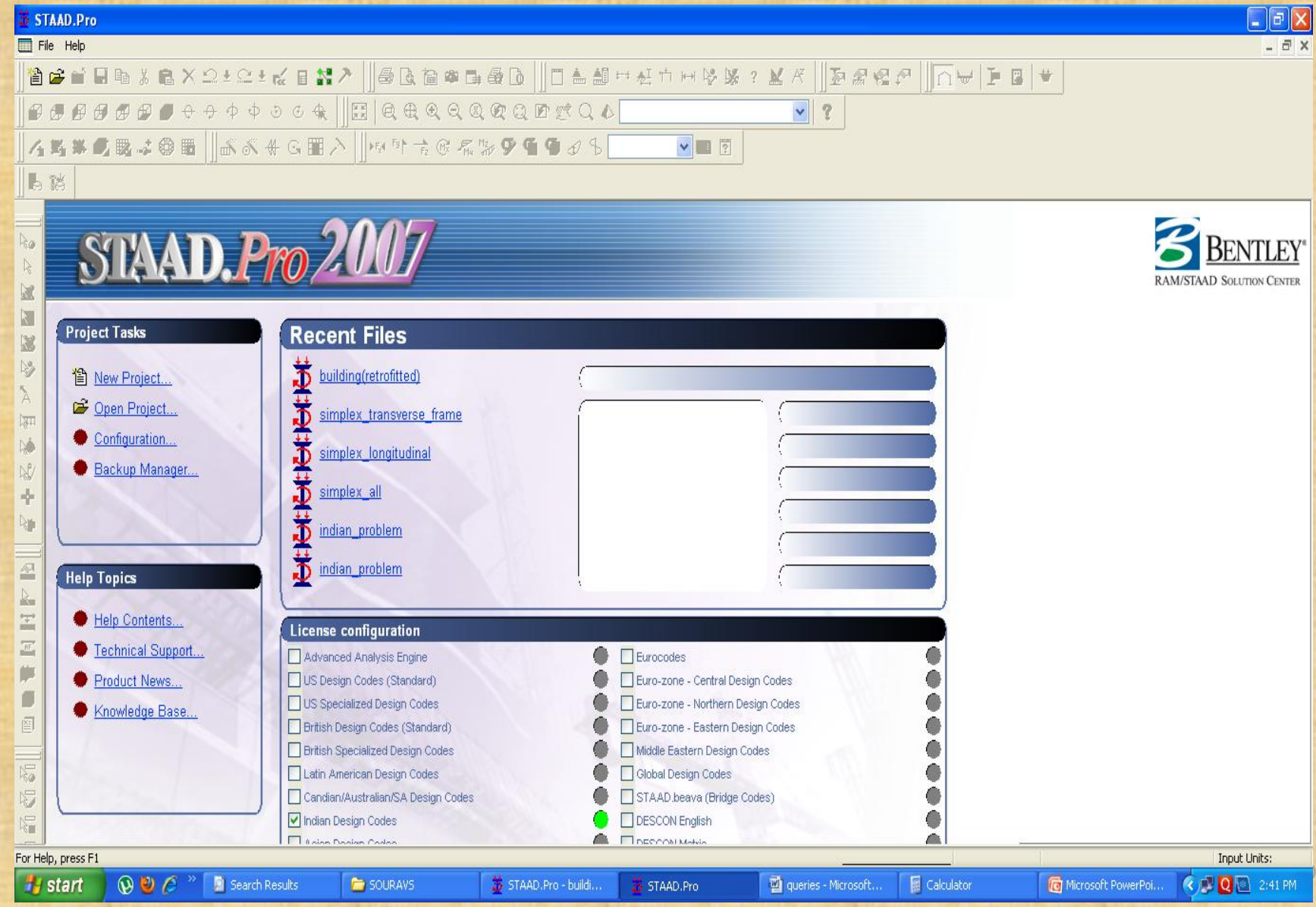



A TYPICAL RCC FRAMED STRUCTURE

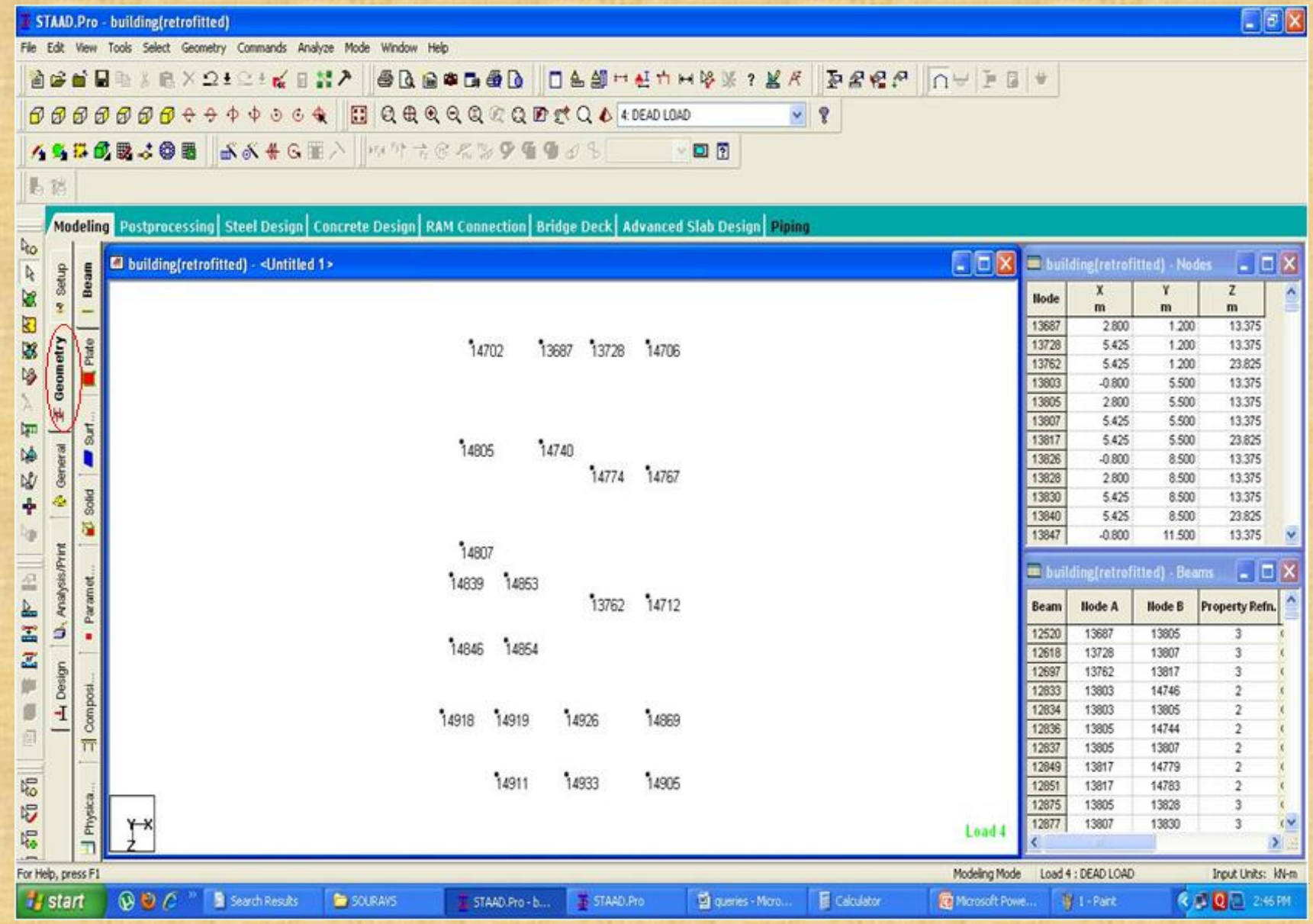



A TYPICAL STEEL TRUSS STRUCTURE

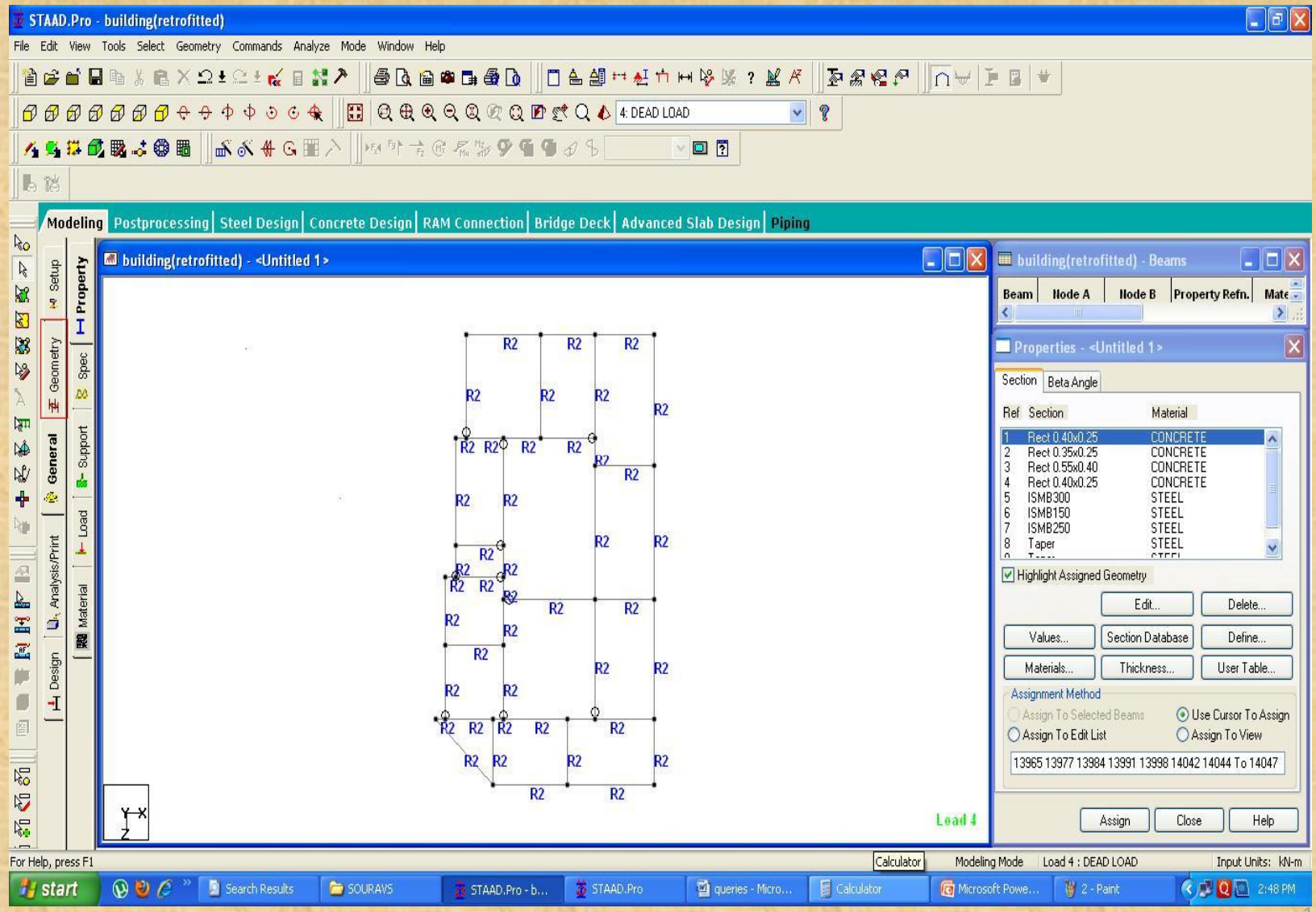
# 1) Create New file or open a previous file



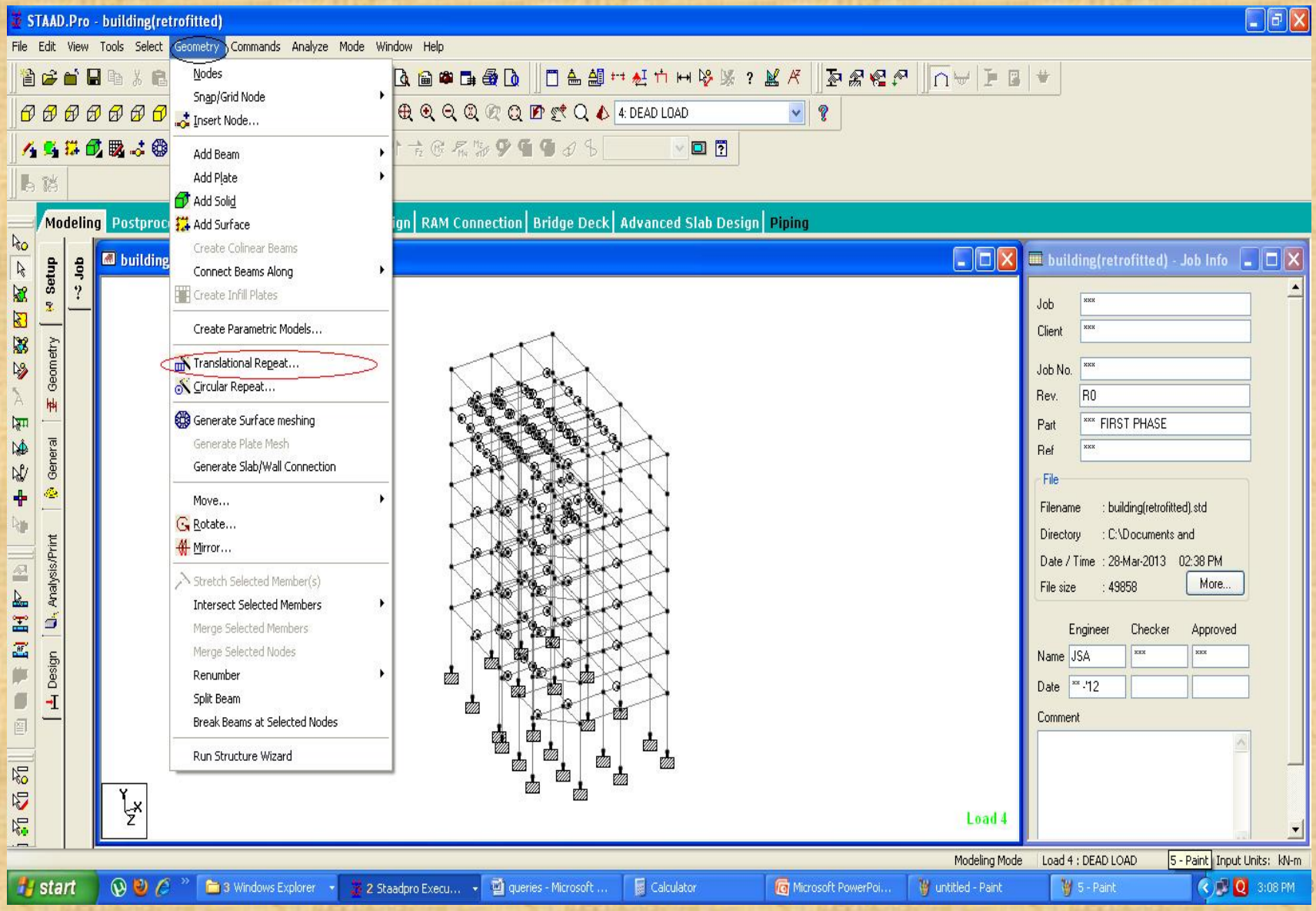
# 2) Create Nodes



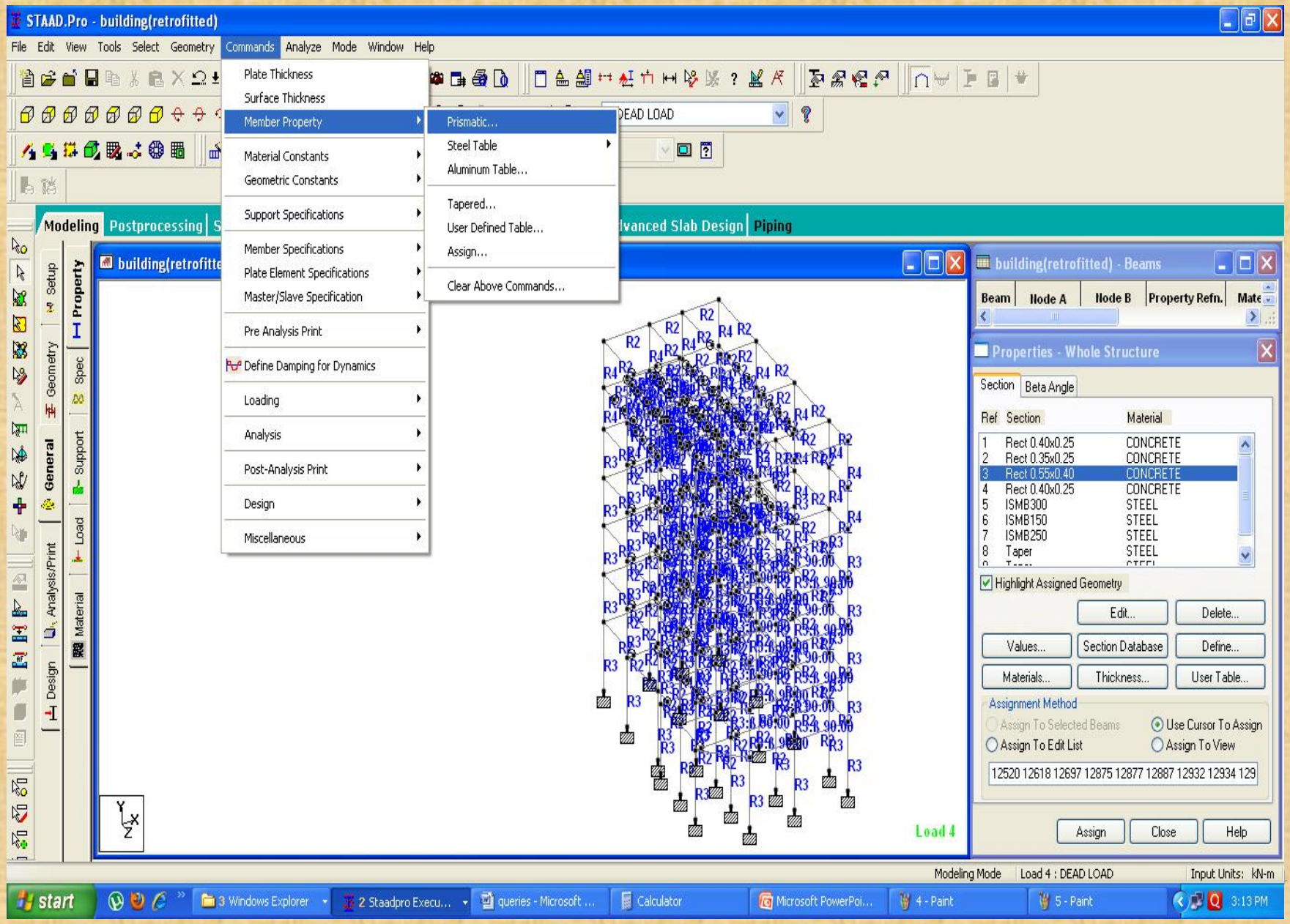
# 3) Add Beams or Plates etc



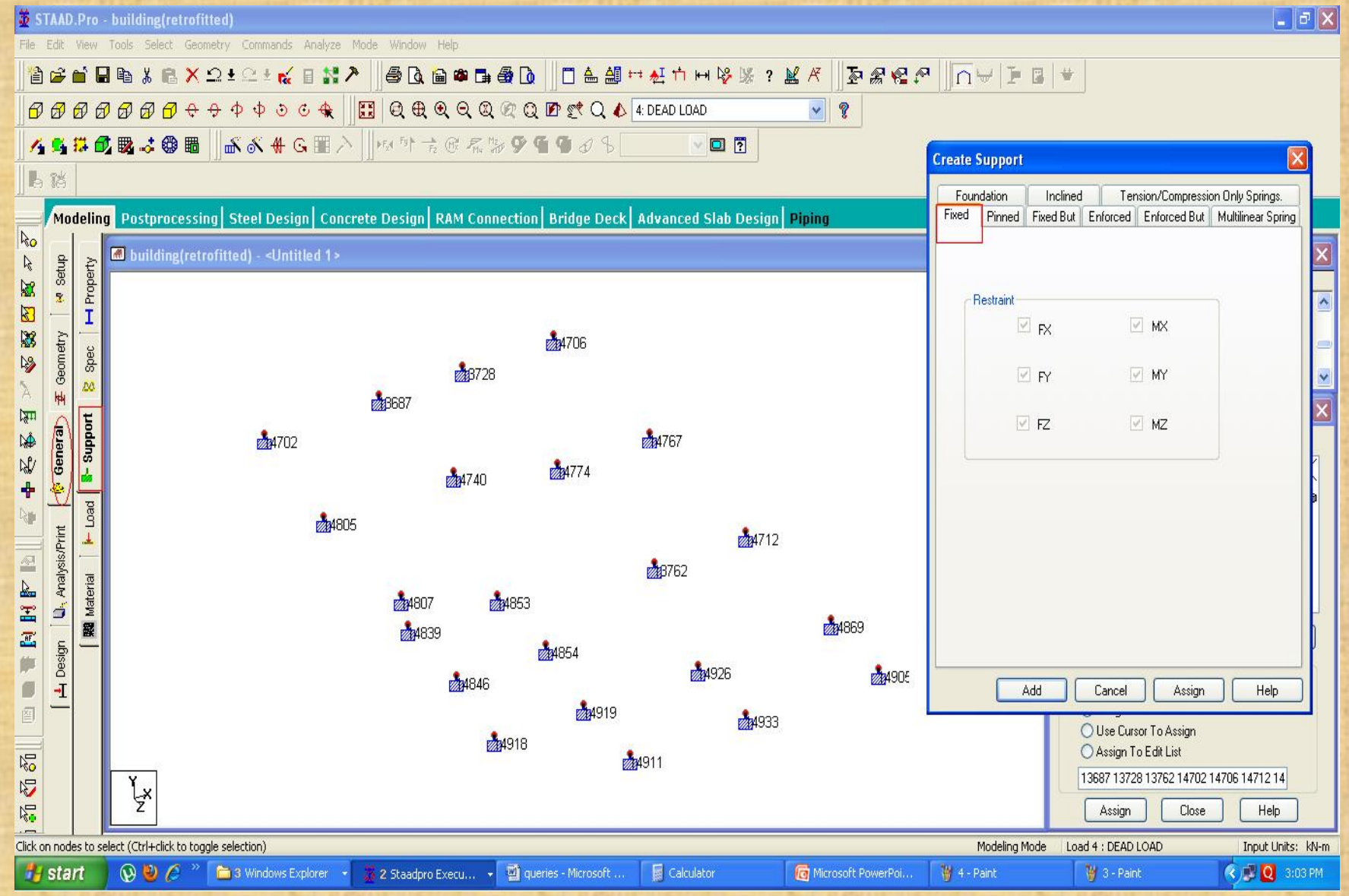
# 4) Create whole structure



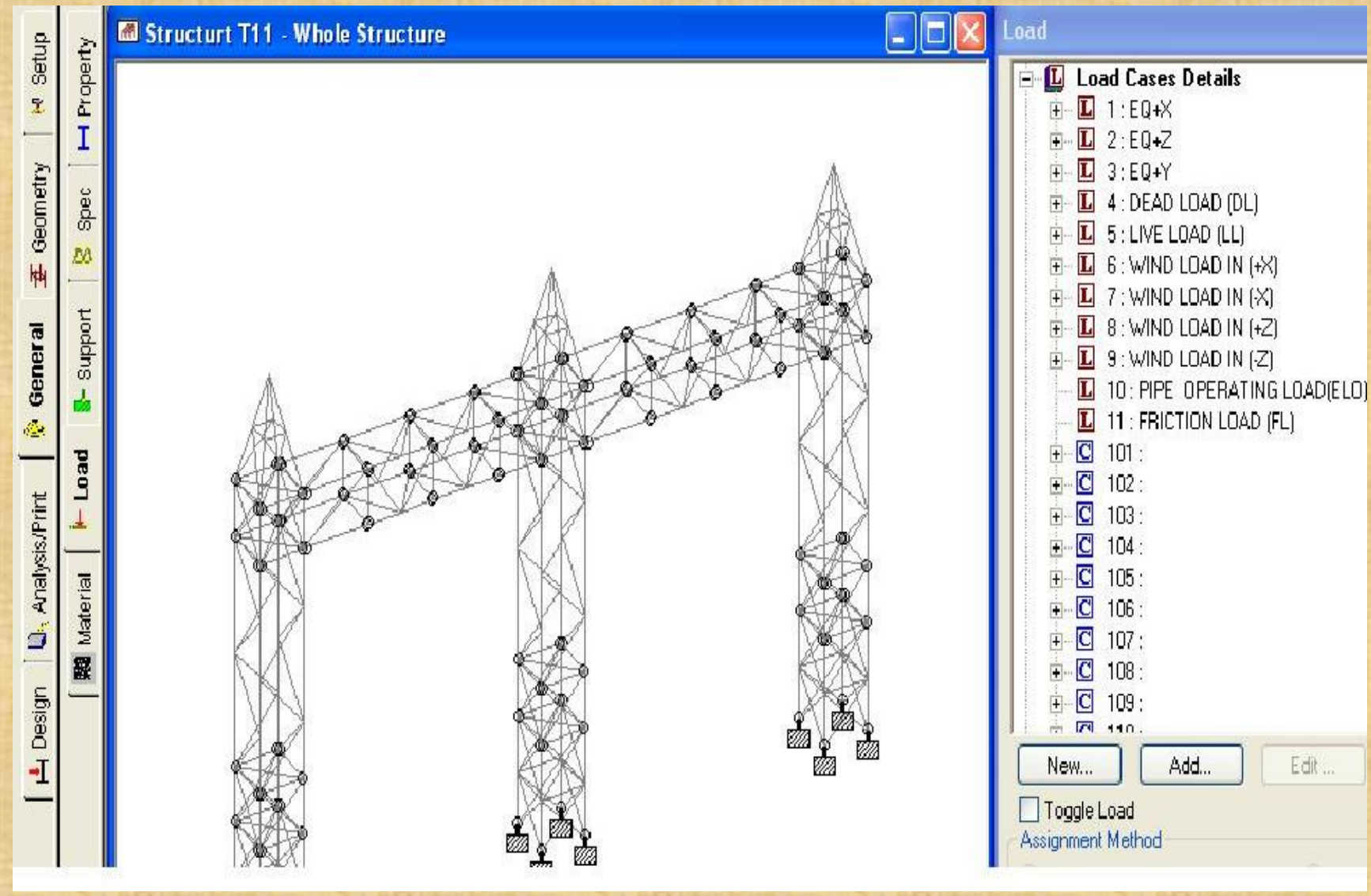
# 5)Assign Material Properties



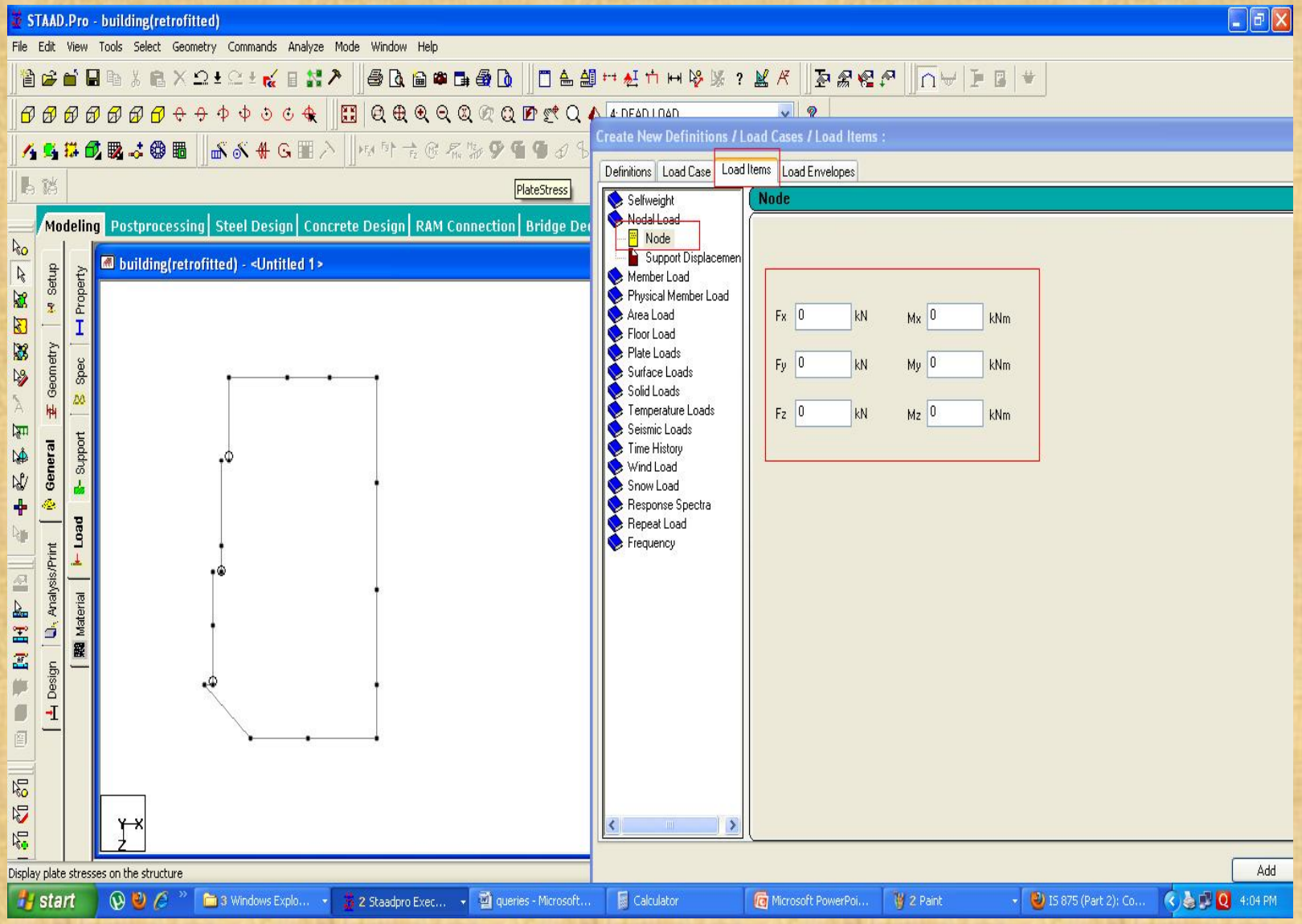
# 6) Add Boundary Conditions



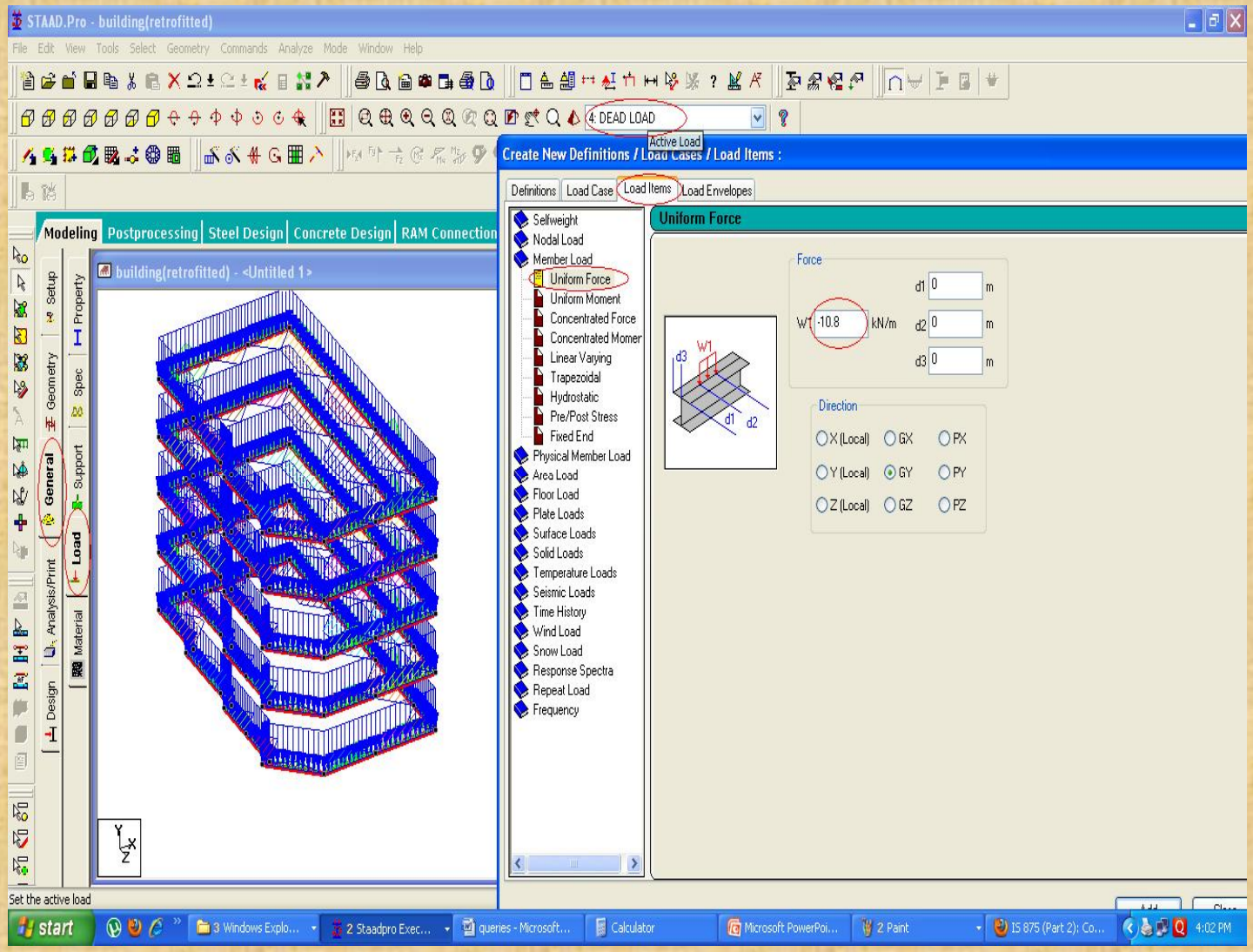
# Basic Loads and combination Generation



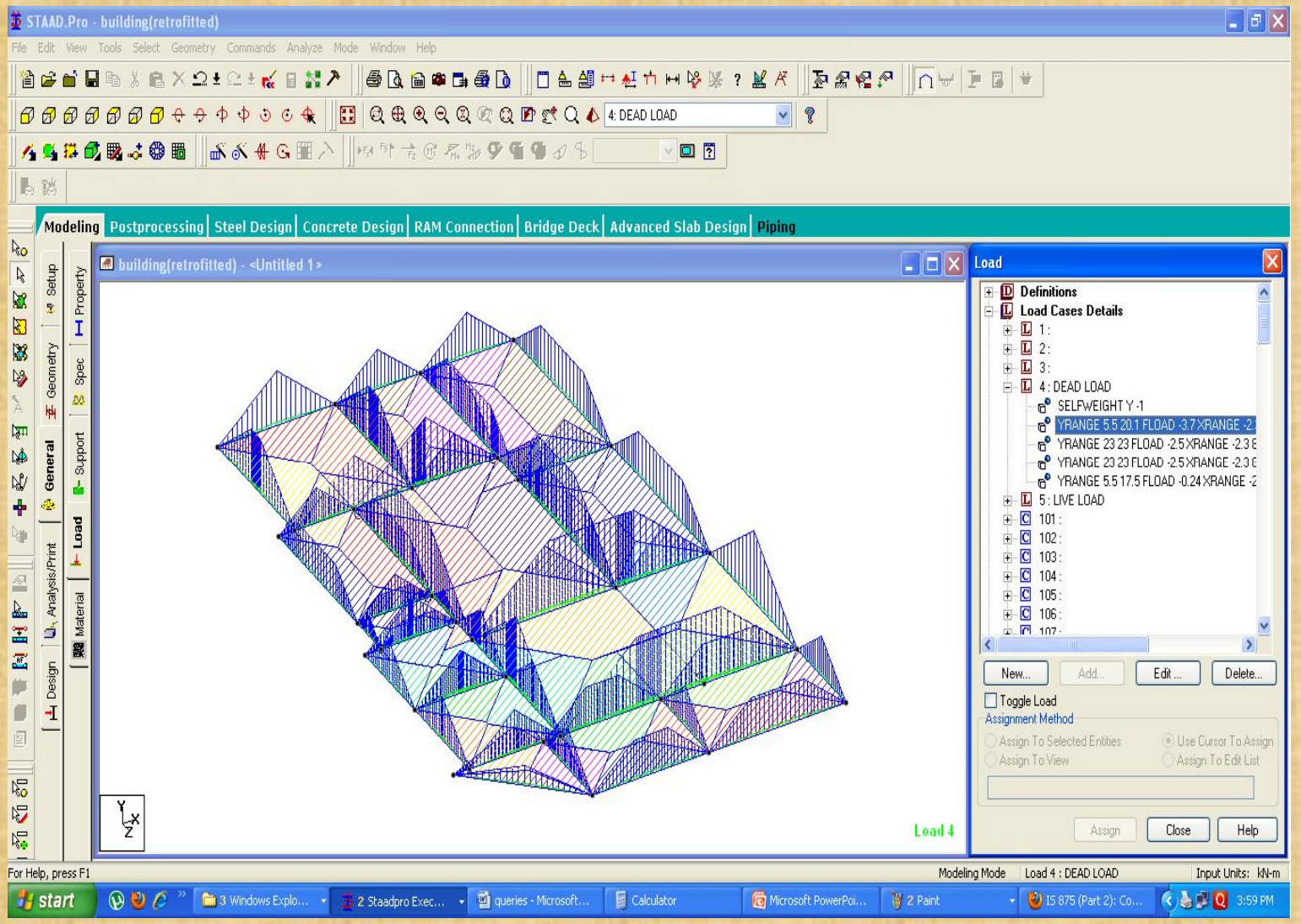
# Creation of Nodal Load



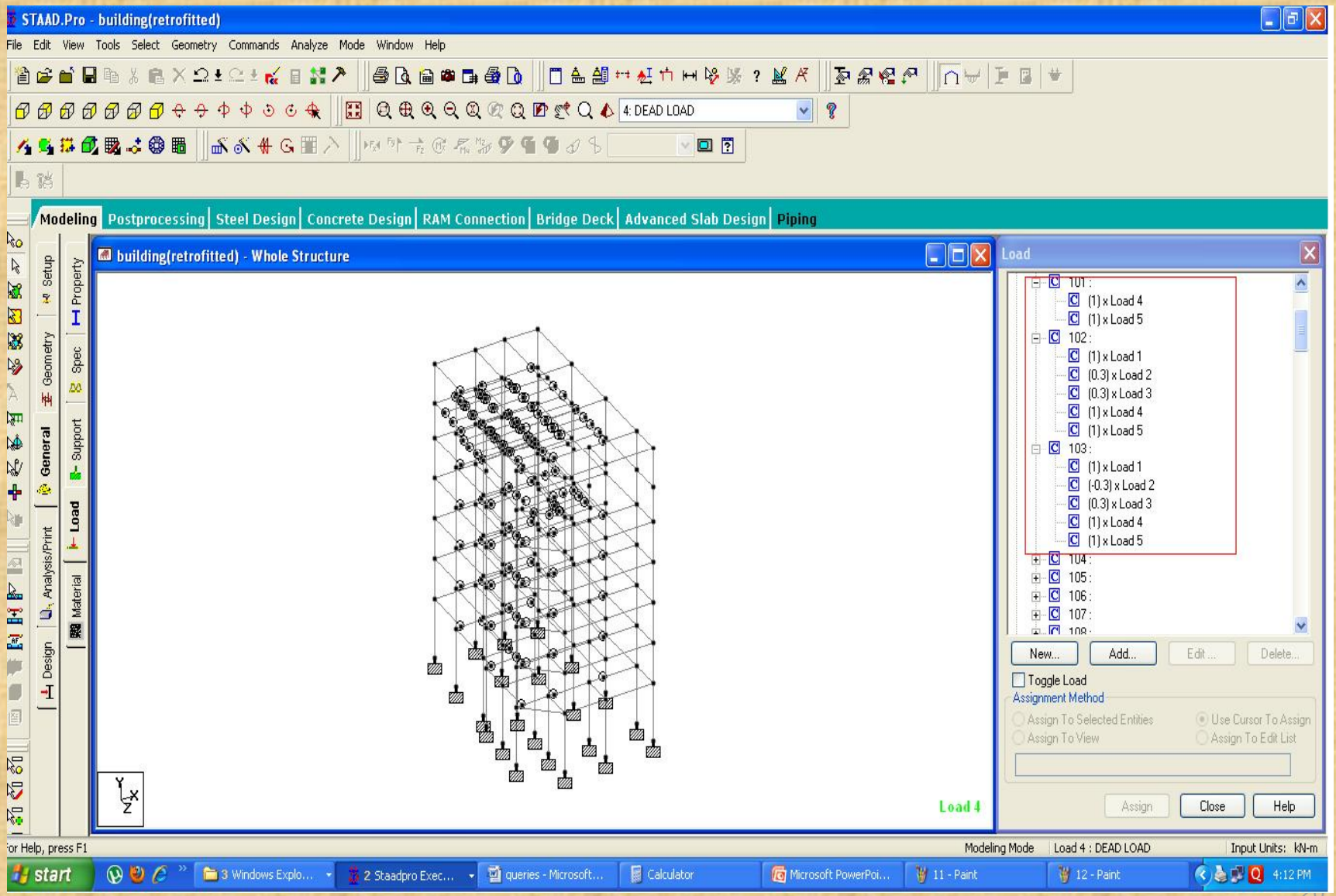
# Creation of UDL



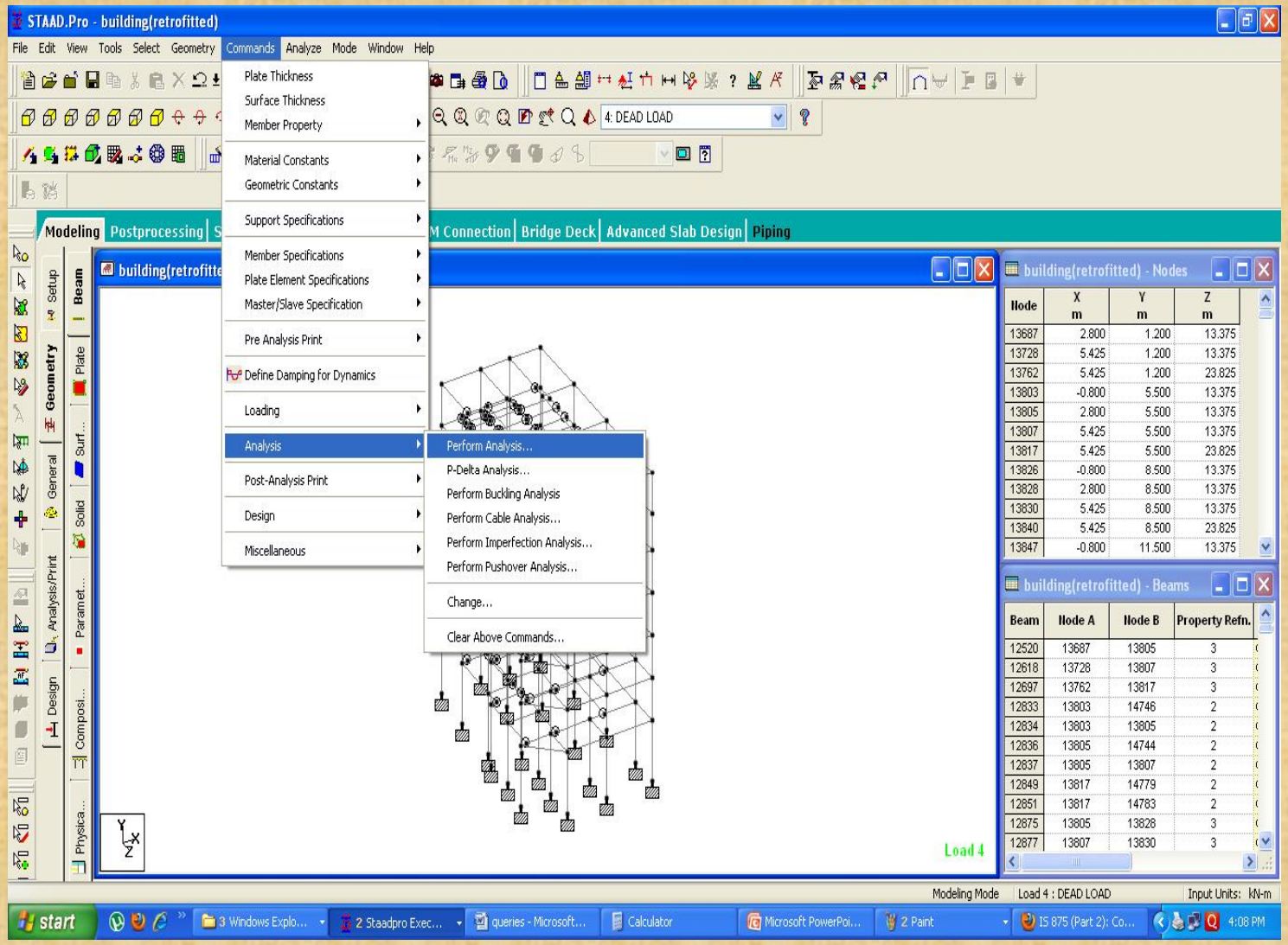
# Creation of Floor Loads



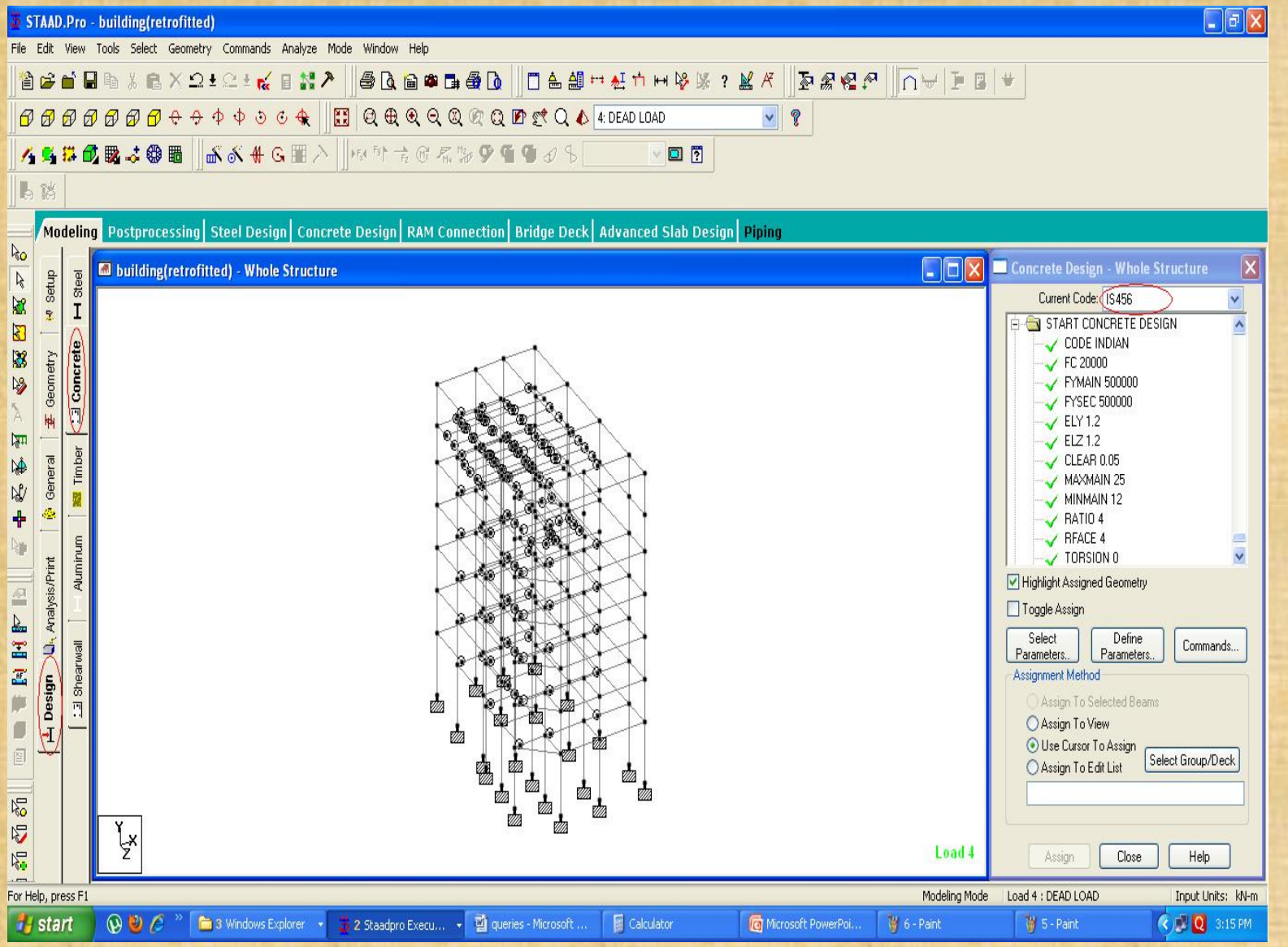
# Creation of Load Combinations



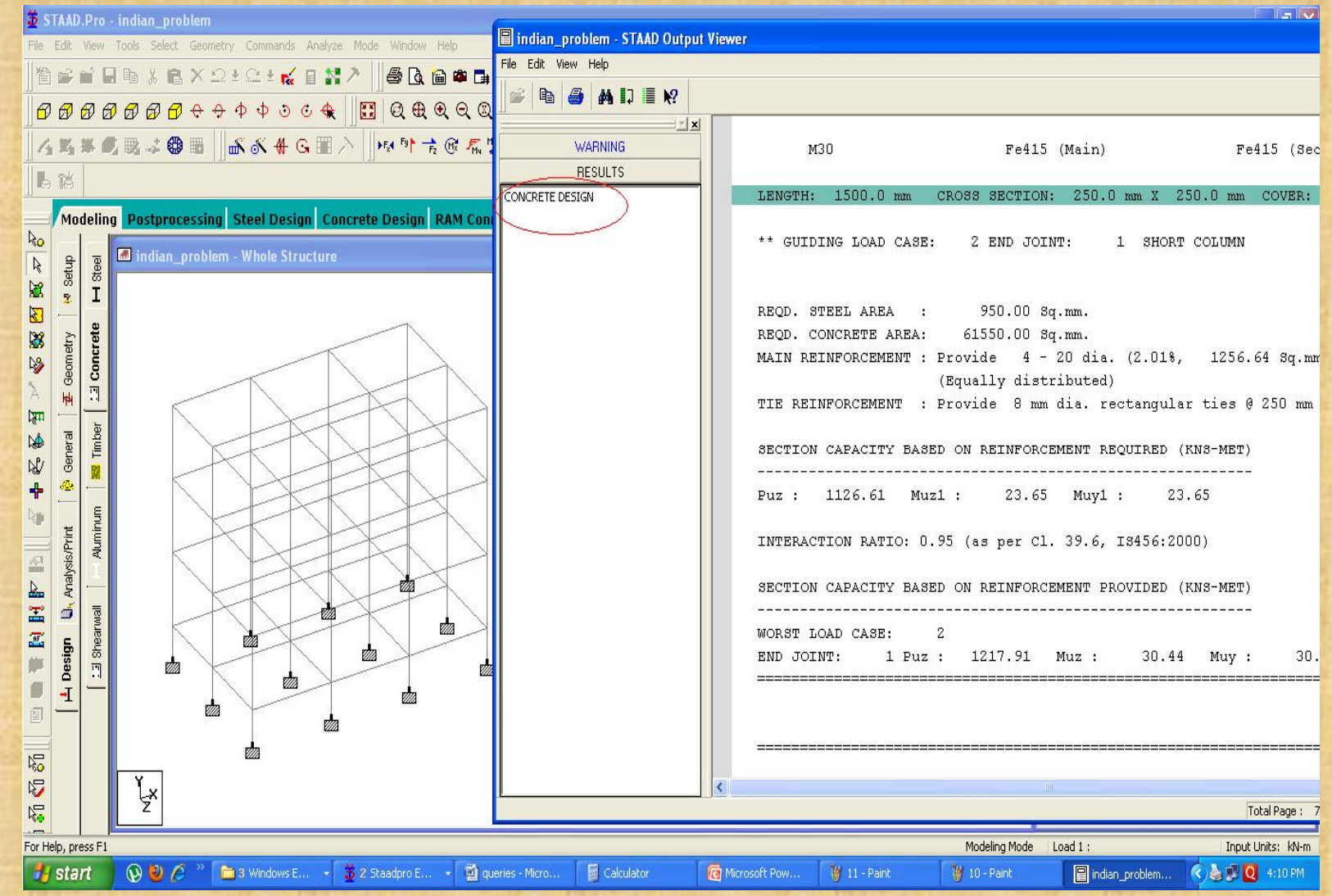
# Analysis Command of Structure



# Design with proper Design Command



# Typical Design Output File



# THANK YOU