RFEM

Aalto University
School of Engineering

Finite Element Method (FEM)

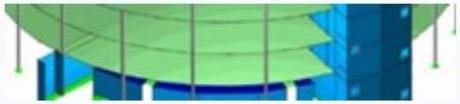
- A numerical method for finding an approximation for a phenomenon described by partial differential equations and boundary conditions.
- The object is divided into finite elements that work together.

Finite Element Method (FEM)

- Efficiency is based on flexibility: complex shape, materials, support conditions and loads.
- Solved in practice by a computer program.
- Approximation method - the solution includes an error in the method.

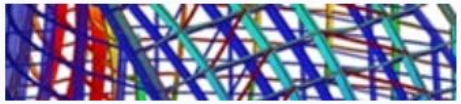
Dlubal Products

RFEM - FEA SOFTWARE



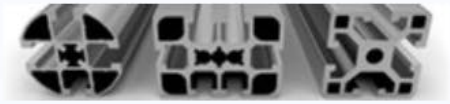
- RFEM - 3D-FEA Program
- RFEM Add-on Modules
  - Reinforced Concrete Structures
  - Steel and Aluminium Structures
  - Timber Structures
  - Glass Structures
  - Towers and Masts
  - Connections
  - Dynamic Analysis
  - Piping Systems
  - Tensile Membrane Structures
  - Others

RSTAB - FRAMES & TRUSSES



- RSTAB - Frame Analysis
- RSTAB Add-on Modules
  - Reinforced Concrete Structures
  - Steel and Aluminium Structures
  - Timber Structures
  - Towers and Masts
  - Connections
  - Dynamic Analysis
  - Others

CROSS-SECTION PROPERTIES



- SHAPE-THIN - Thin-Walled Cross-Sections
- SHAPE-MASSIVE - Thick-Walled Cross-Sections

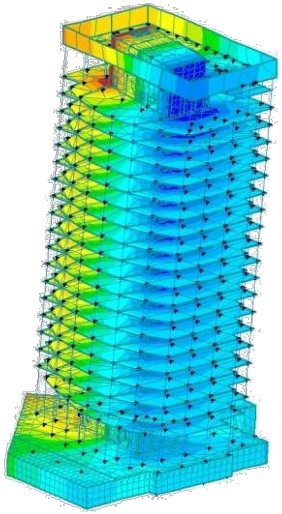
STAND-ALONE PROGRAMS

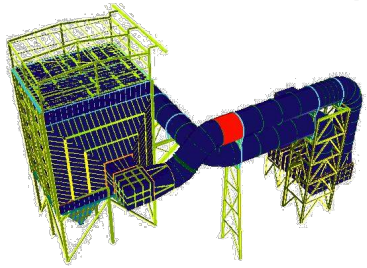
- Steel Structures
  - Craneway Girder Design
  - Plate Buckling Analysis
- Composite Structures
  - Composite Beams
- Timber Structures
  - Glued-Laminated Beams
  - Continuous Beams
  - Columns
  - Purlins
  - Three-Hinged Frames
  - Stiffening Truss Bracings
  - Roofs

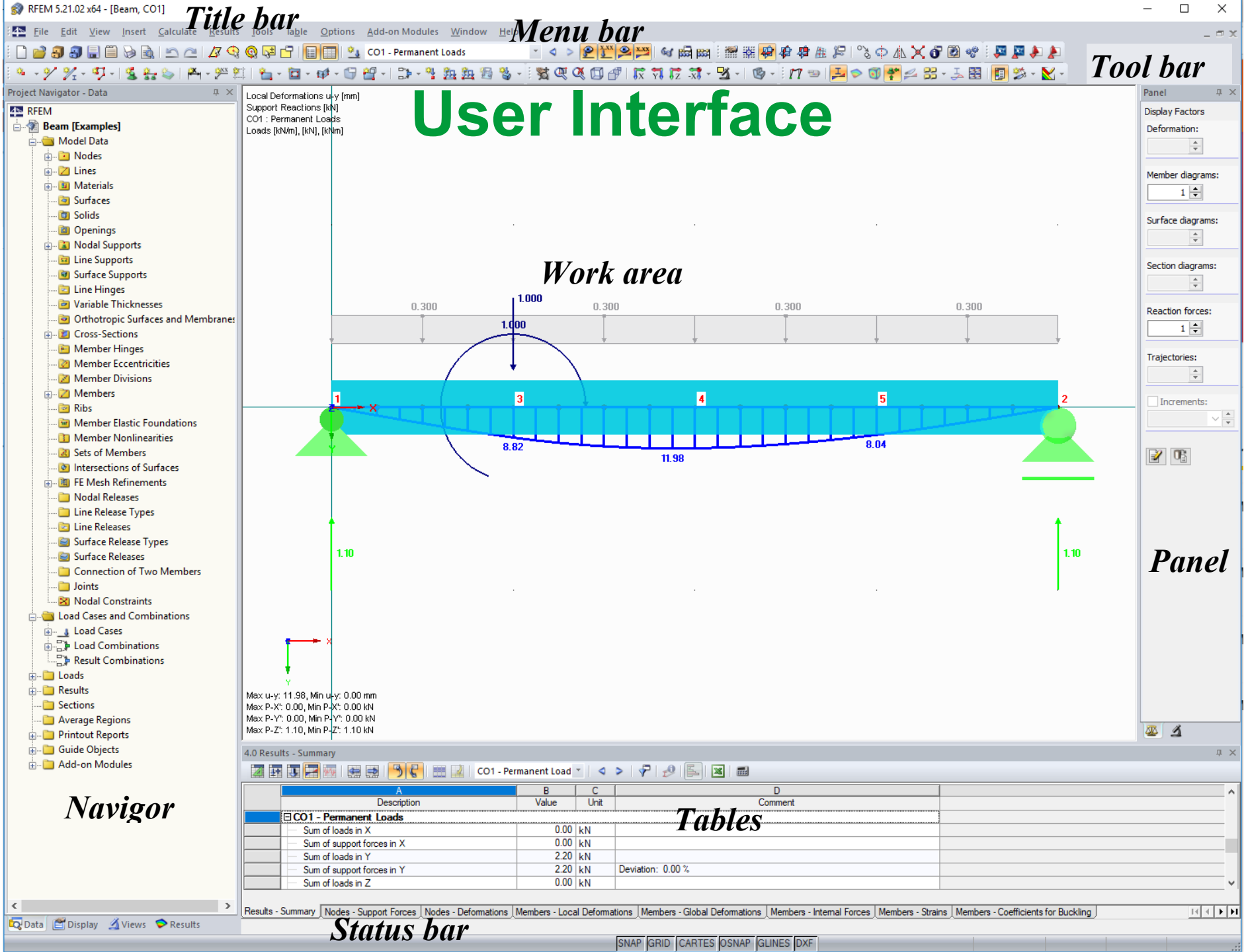
https://www.dlubal.com/en

RFEM

- The program can be used extensively in construction-related design:
  - Houses, skyscrapers, office buildings
  - Bridges, tunnels, railways
  - Industrial construction (warehouses, halls)
  - Industry (pressure boilers, pipelines)
  - Water and environment (water treatment plants, etc.)

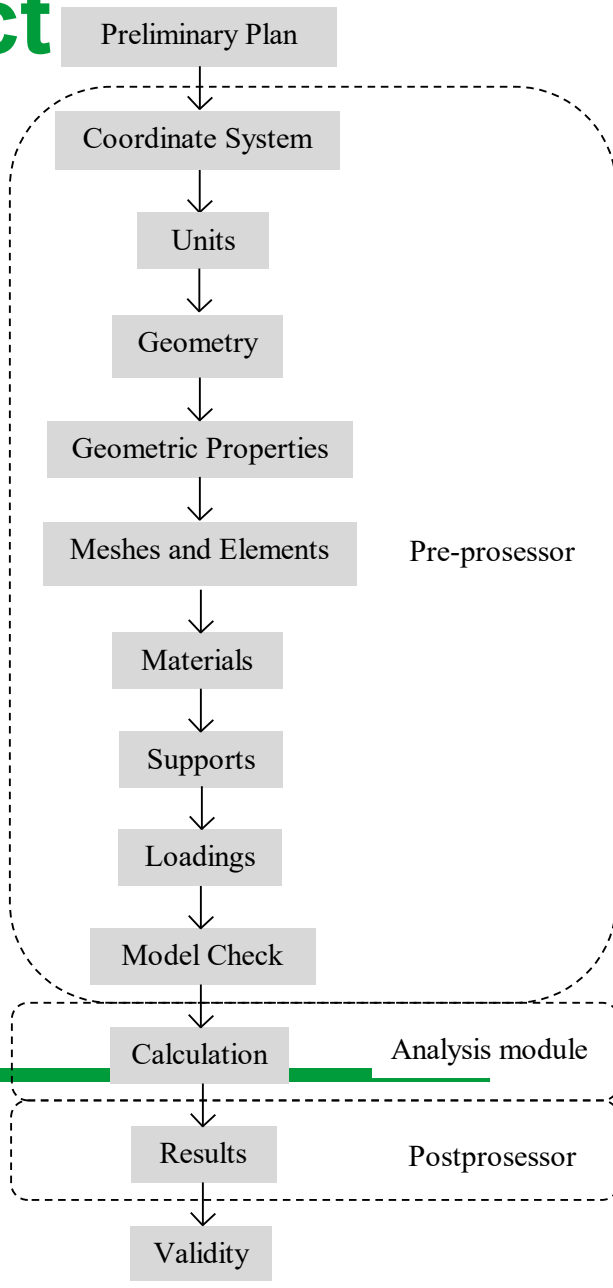





Title bar
Menu bar
Tool bar
User Interface
Work area
Panel
Navigor
Tables
Status bar
RFEM 5.21.02 x64 - [Beam, CO1]
File Edit View Insert Calculate Results Tools Table Options Add-on Modules Window Help
CO1 - Permanent Loads
Project Navigator - Data
RFEM
Beam [Examples]
Model Data
Nodes
Lines
Materials
Surfaces
Solids
Openings
Nodal Supports
Line Supports
Surface Supports
Line Hinges
Variable Thicknesses
Orthotropic Surfaces and Membranes
Cross-Sections
Member Hinges
Member Eccentricities
Member Divisions
Members
Ribs
Member Elastic Foundations
Member Nonlinearities
Sets of Members
Intersections of Surfaces
FE Mesh Refinements
Nodal Releases
Line Release Types
Line Releases
Surface Release Types
Surface Releases
Connection of Two Members
Joints
Nodal Constraints
Load Cases and Combinations
Load Cases
Load Combinations
Result Combinations
Loads
Results
Sections
Average Regions
Printout Reports
Guide Objects
Add-on Modules
Data Display Views Results
Local Deformations u-y [mm]
Support Reactions [kN]
CO1 : Permanent Loads
Loads [kN/m], [kN], [kNm]
0.300
1.000
1.000
0.300
0.300
0.300
1
2
3
4
5
8.82
11.98
8.04
1.10
1.10
Max u-y: 11.98, Min u-y: 0.00 mm
Max P-X': 0.00, Min P-X': 0.00 kN
Max P-Y': 0.00, Min P-Y': 0.00 kN
Max P-Z': 1.10, Min P-Z': 1.10 kN
Panel
Display Factors
Deformation:
Member diagrams:
Surface diagrams:
Section diagrams:
Reaction forces:
Trajectories:
Increments:
4.0 Results - Summary
CO1 - Permanent Load
A Description
B Value
C Unit
D Comment
CO1 - Permanent Loads
Sum of loads in X 0.00 kN
Sum of support forces in X 0.00 kN
Sum of loads in Y 2.20 kN
Sum of support forces in Y 2.20 kN Deviation: 0.00 %
Sum of loads in Z 0.00 kN
Results - Summary
Nodes - Support Forces
Nodes - Deformations
Members - Local Deformations
Members - Global Deformations
Members - Internal Forces
Members - Strains
Members - Coefficients for Buckling
SNAP GRID CARTES OSNAP GLINES DXF

Modelling Project

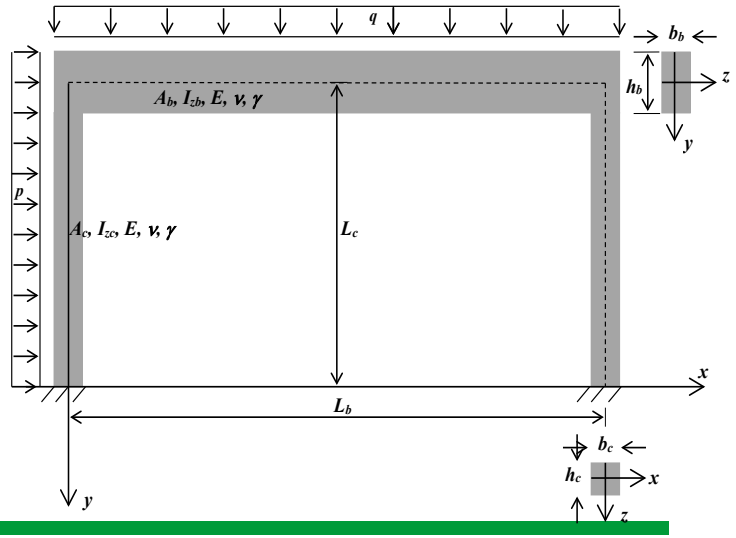
1. Preliminary plan
2. Coordinate system
3. Units
4. Geometry
5. Geometric properties of the cross-section
6. Elements and meshes
7. Materials
8. Supports
9. Loading
10. Model check
11. Analysis
12. Results
13. Validity
14. Documentation



1. Preliminary Planning

Planned before modeling with the program:

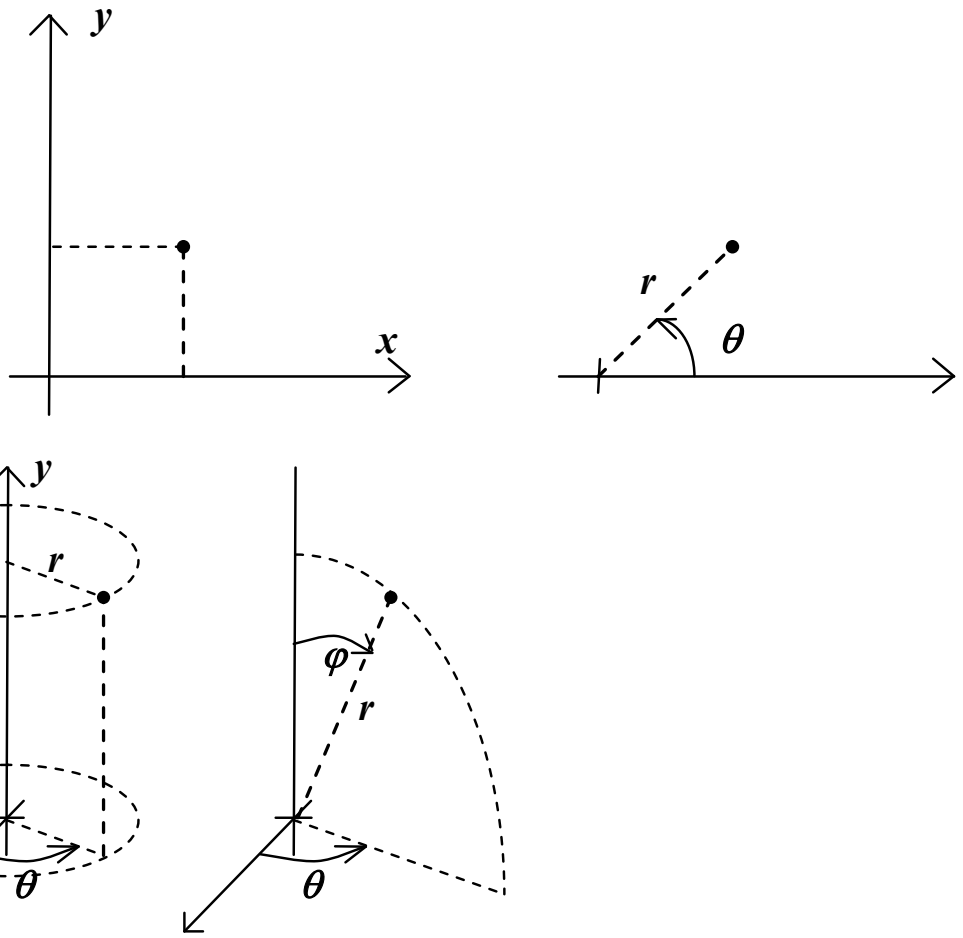
1) Geometry and geometric properties of the cross-sections
2) Coordinate system
3) Supports (boundary conditions)
4) Materials: models ja parameters
5) Loading and load combination
6) Types of structures
7) Elements and meshes
8) Units
9) Design criteria (Eurocodes)
10) Analysing criteria
    - static or dynamic
    - linear or nonlinear



2. Coordinate System

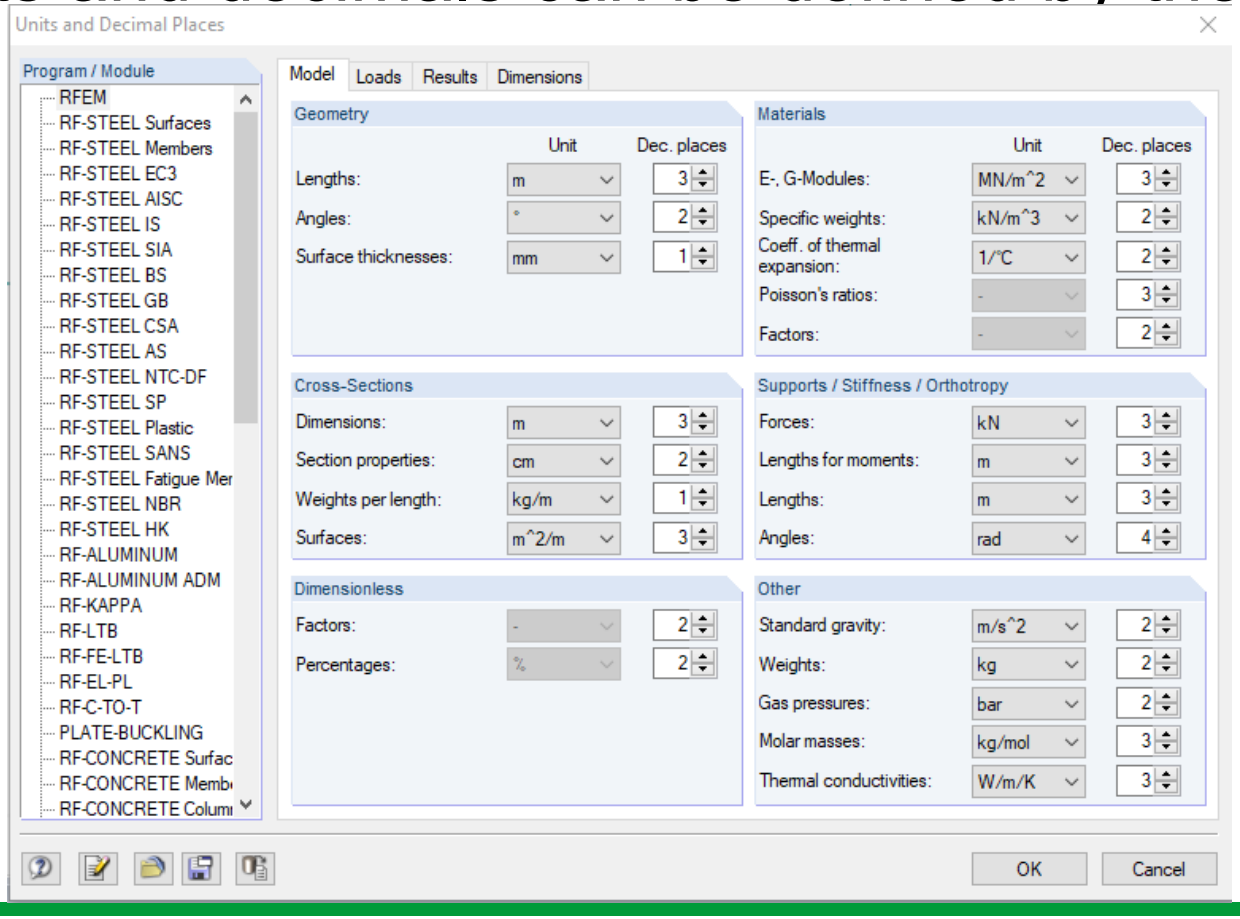
2D and 3D:

- Cartesian
- Polar or cylindrical
- Spherical



3. Units

- Units and decimals can be defined by the user.

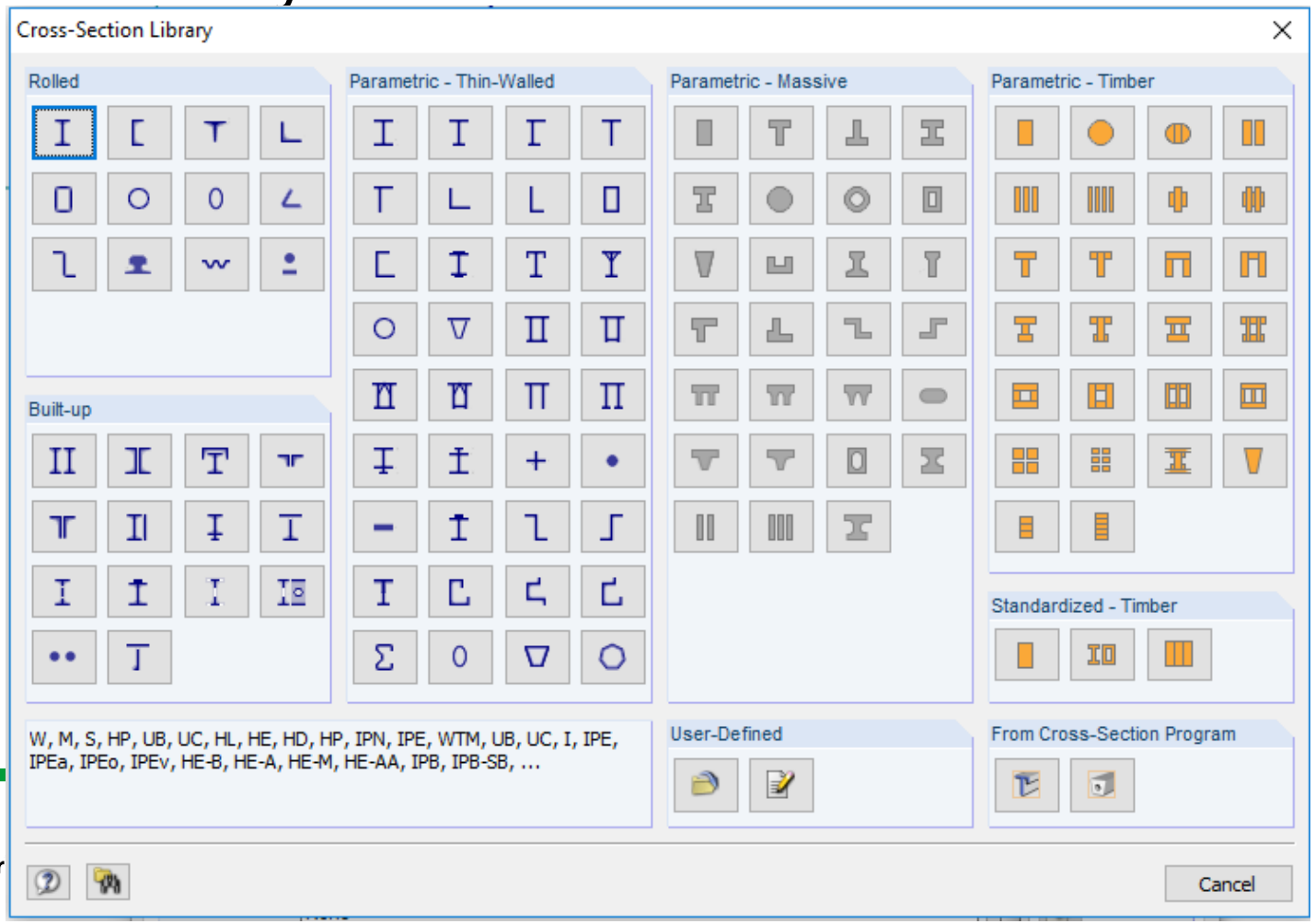


4. Geometry

- Creating a model with a preprocessor.
- Import geometry created with another program into RFEM.
  - AutoCad (DWG)
  - Tekla Structures (IFC, HT B4)
- Parametric modeling.

5. Geometric Properties of the Cross-Section

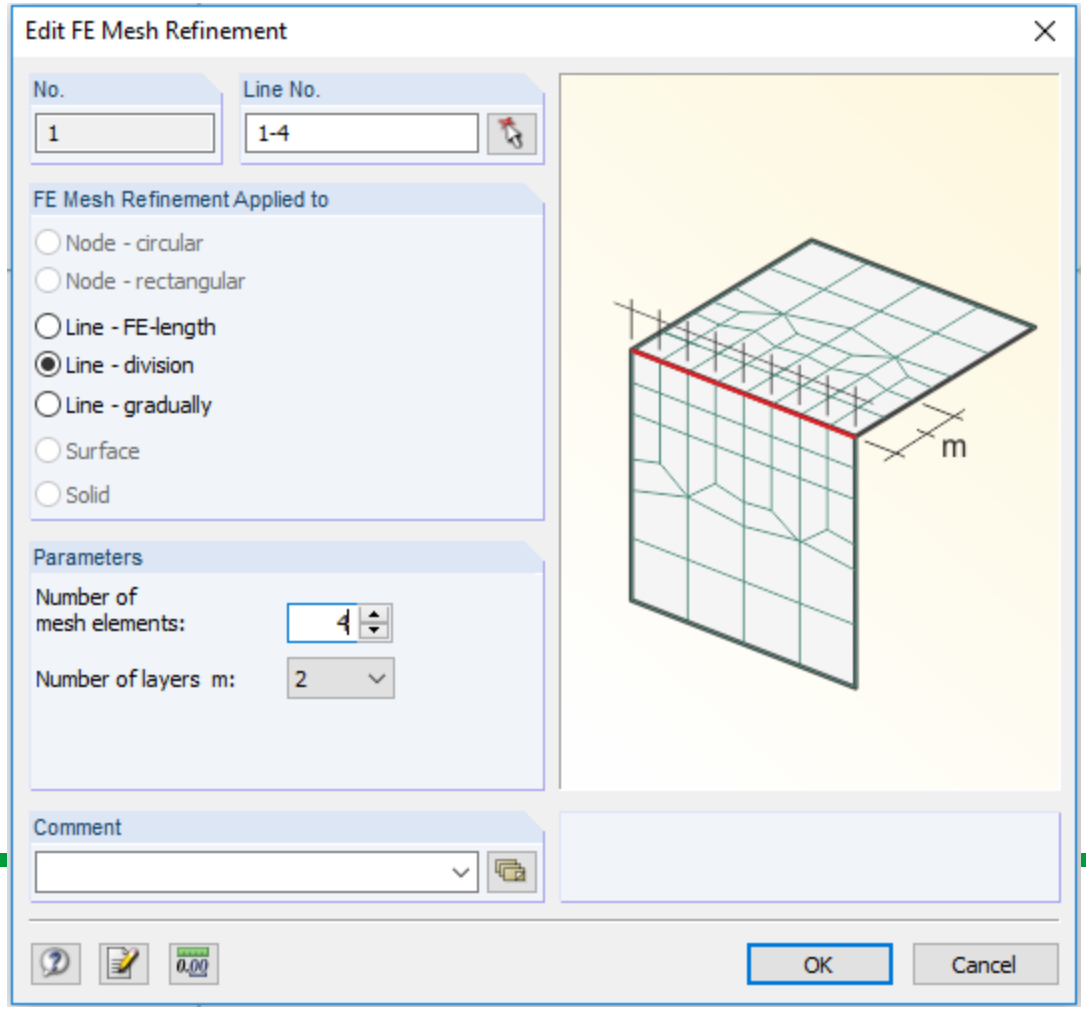
Cross-section library



6. Elements and Meshes

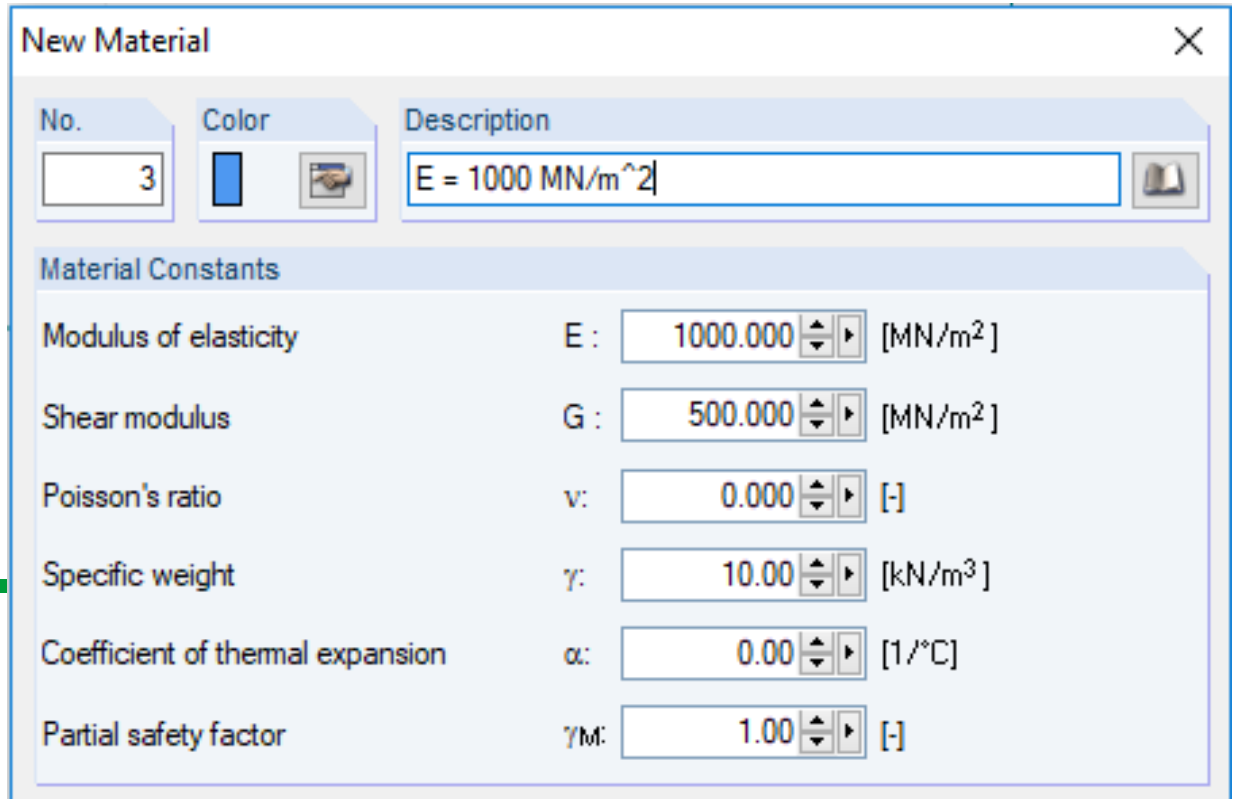
Element types by structure types

- Cable
- Bar
- Beam
- Plane plate
- Slab plate
- Membrane
- Shell



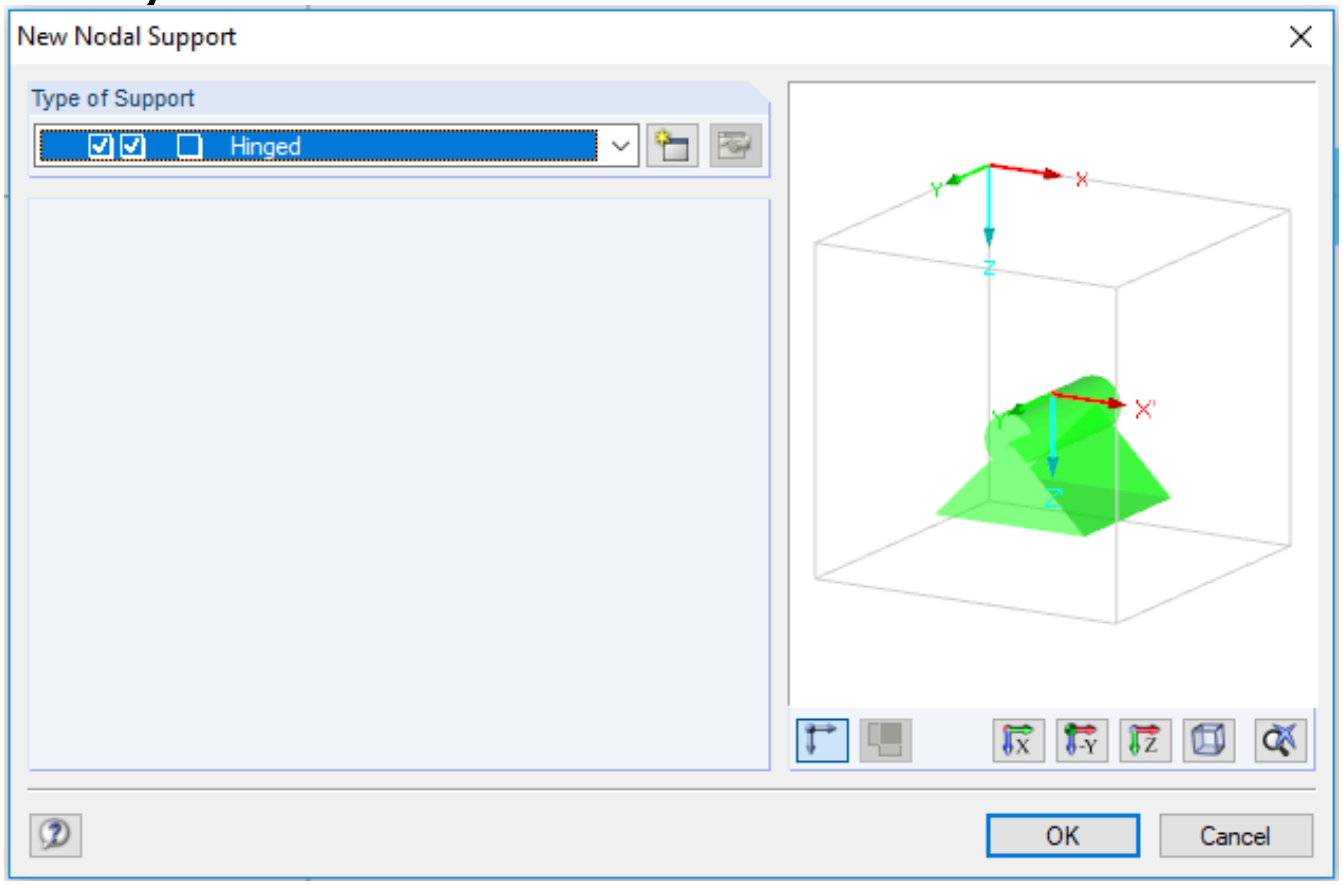
7. Materials

- Material library
- Aluminium, carbon fibre, concrete, glass, iron, steel, stone, timber etc.
- User-defined material parameters.



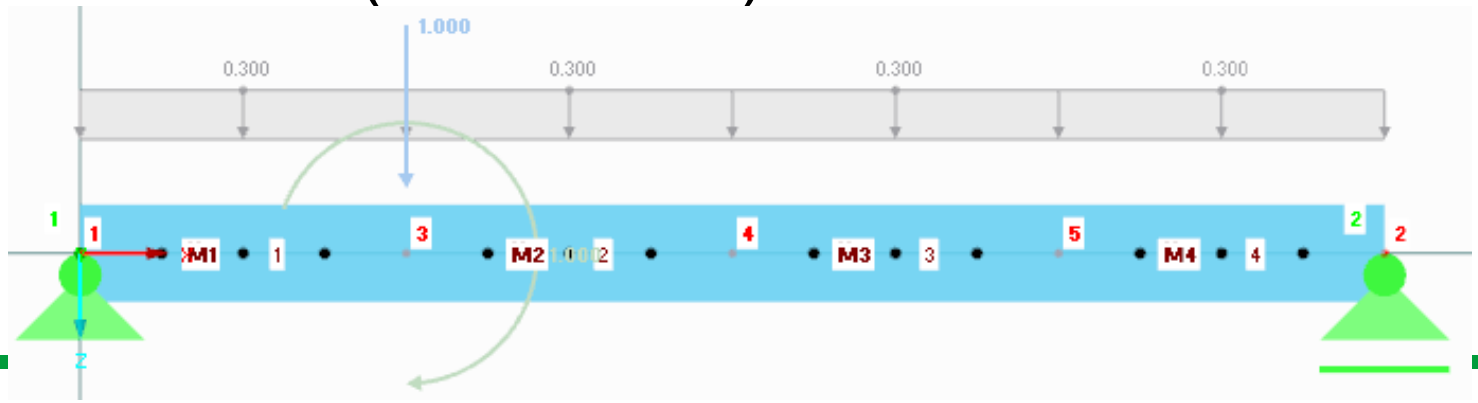
8. Supports

Boundary conditions



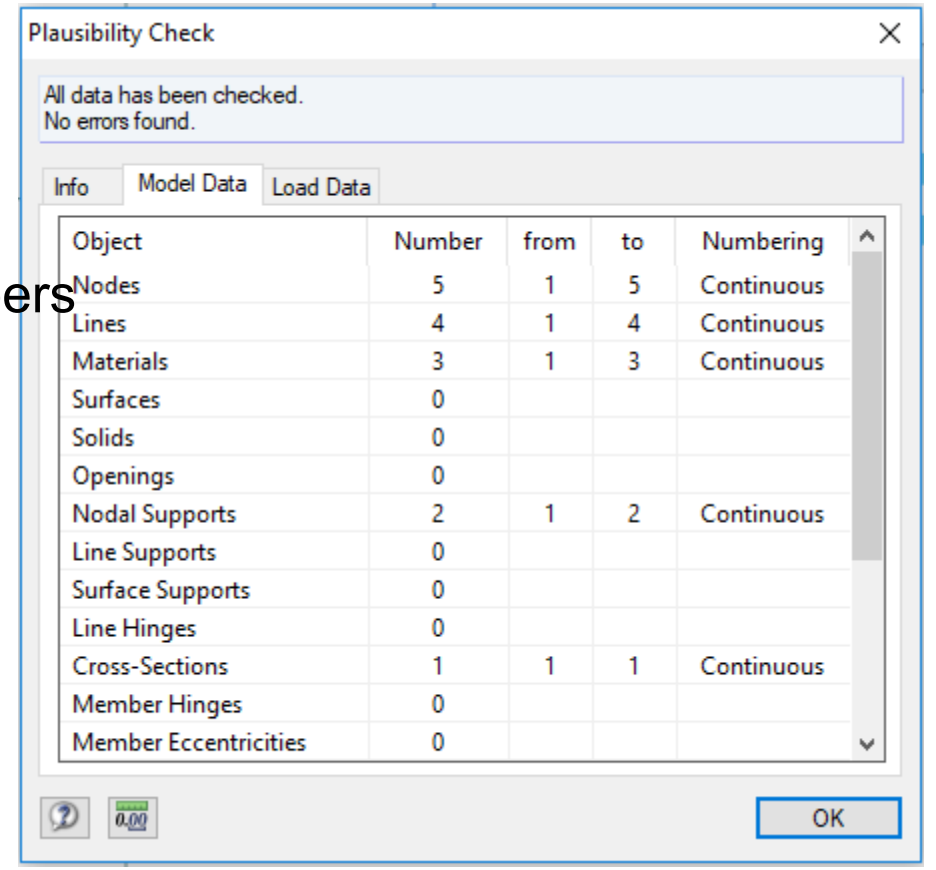
9. Loading

- Static / dynamic loads
- Permanent / variable loads
- Moving loads
- Load cases
- Combining loads
- Standards (Eurocodes) or user-defined coefficients



10. Model Checking

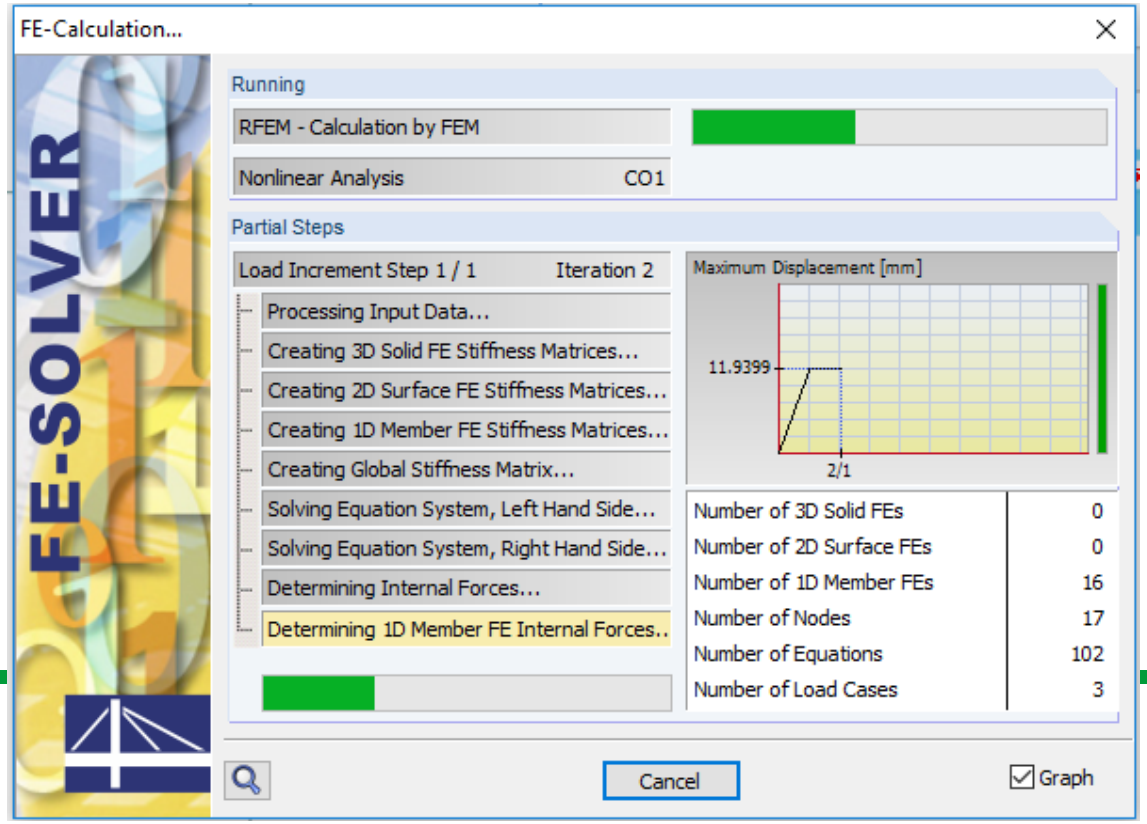
- General checking: necessary information
- Geometric checking
  - identical nodes
  - overlapping members
  - crossing unconnected members
  - overlapping lines
  - crossing unconnected lines
  - unused zero lines
  - overlapping surfaces
  - minimally curved surfaces
- Checking the loads



Object	Number	from	to	Numbering
Nodes	5	1	5	Continuous
Lines	4	1	4	Continuous
Materials	3	1	3	Continuous
Surfaces	0			
Solids	0			
Openings	0			
Nodal Supports	2	1	2	Continuous
Line Supports	0			
Surface Supports	0			
Line Hinges	0			
Cross-Sections	1	1	1	Continuous
Member Hinges	0			
Member Eccentricities	0			

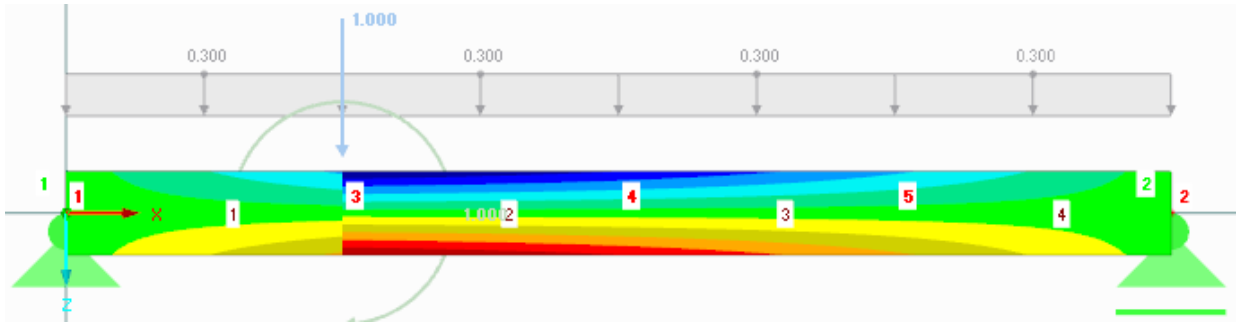
11. Analysis

- Linear, nonlinear
- Static, dynamic
- Steps of a moving load
- Stability
- Earthquake



12. Results

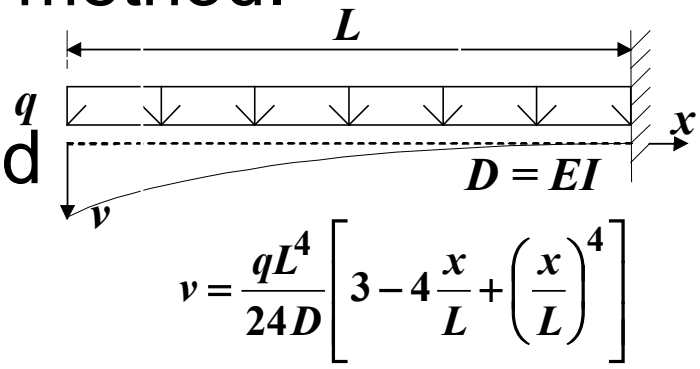
- Deflection
- Rotation
- Axial force
- Shear force
- Moment (bending, torsion)
- Support reactions
- Stress
- Animation



13. Validity

The validity of the results have always been verified by using some other method.

The use of simplified model and manual calculation method is extremely recommended.

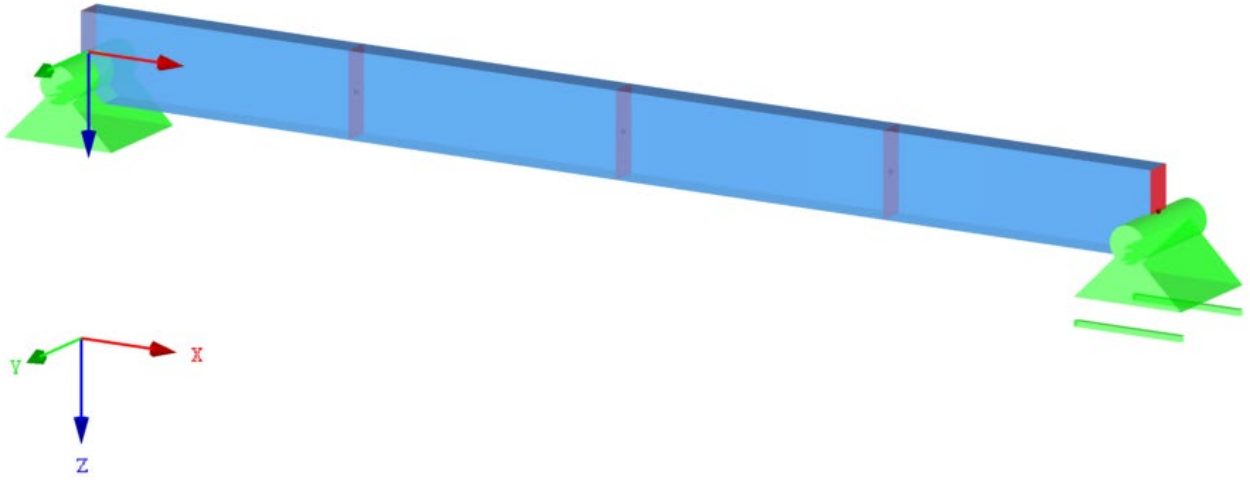


Use RFEM to check the solutions of manual calculation problems in mechanics courses!

14. Documentation

Contents:

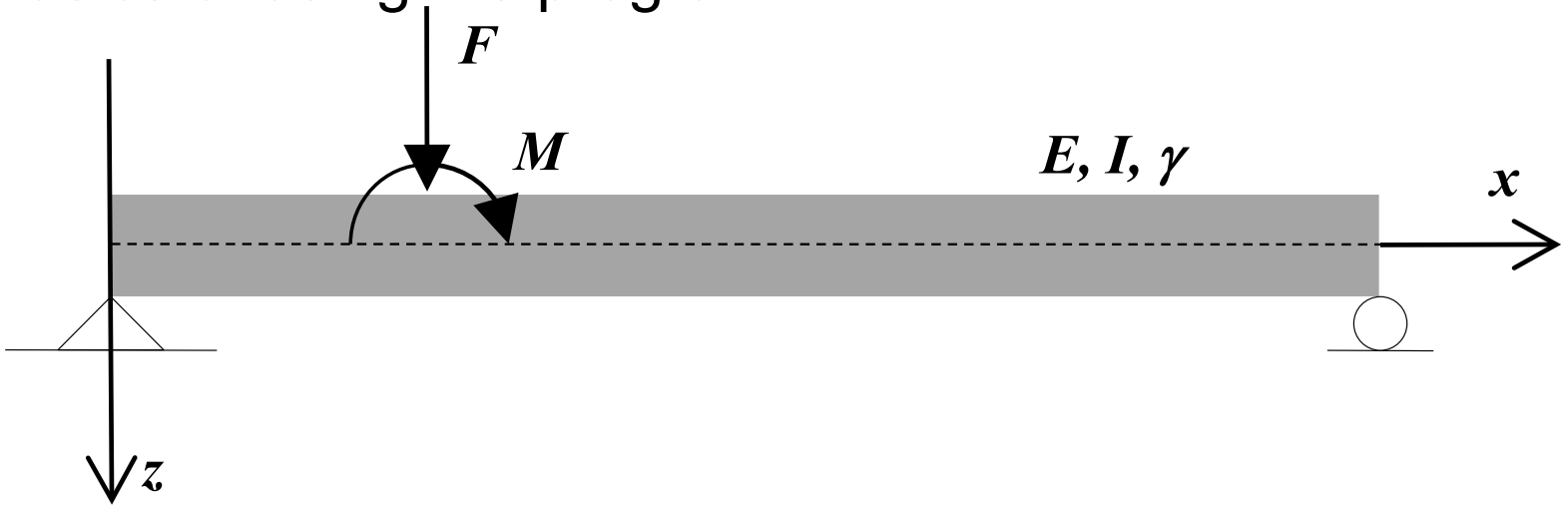
- cover page,
- model figures,
- input and solution data and
- result curves

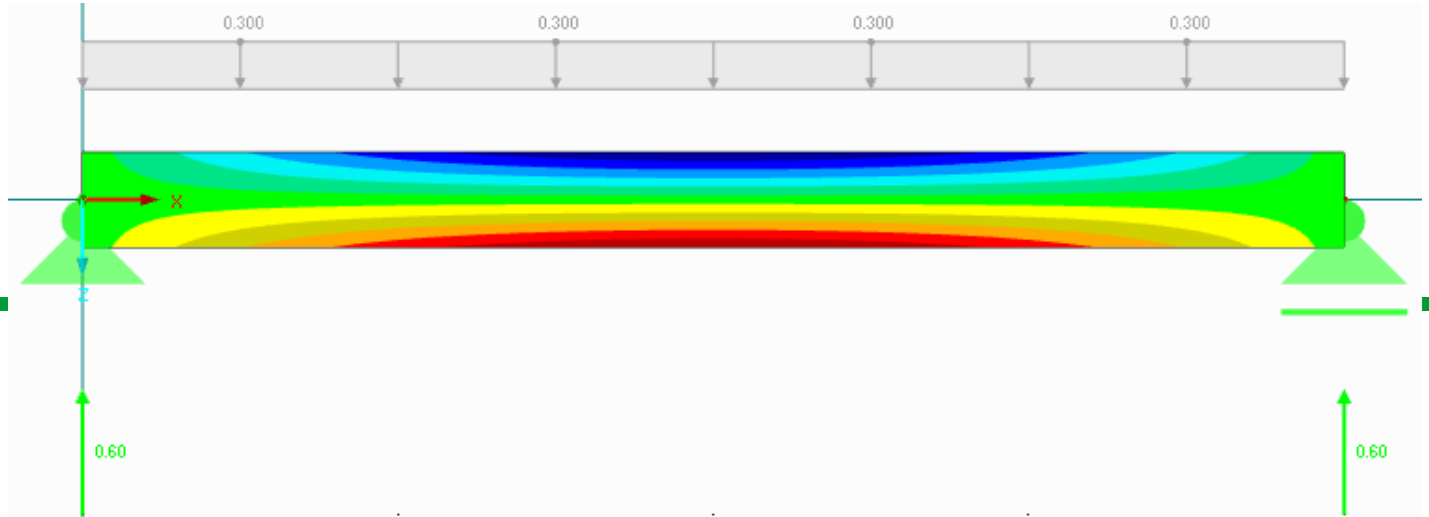


Assignment 5

Simply supported beam

- Basics of using the program

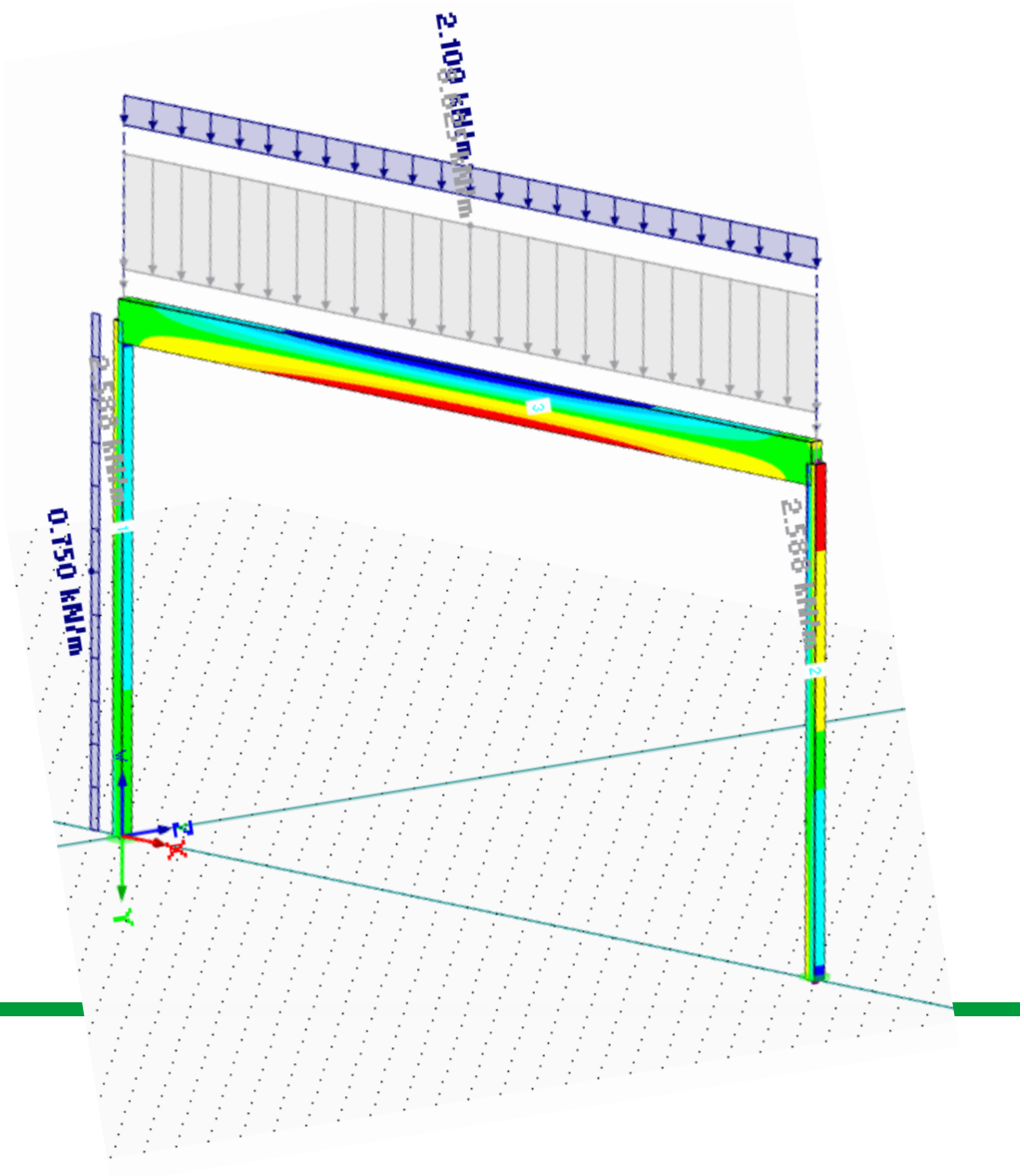




A!

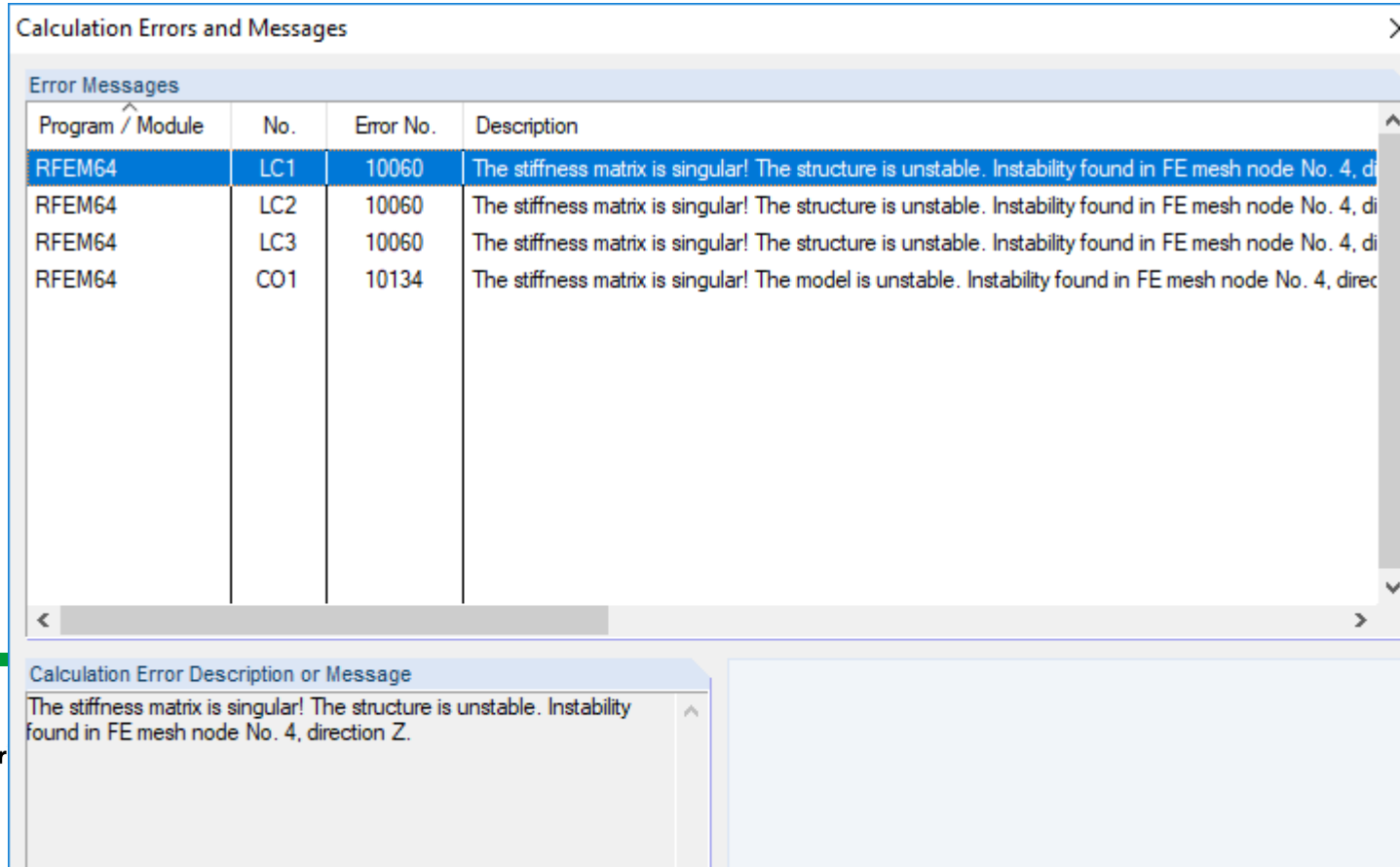
Assingment 6

Concrete frame



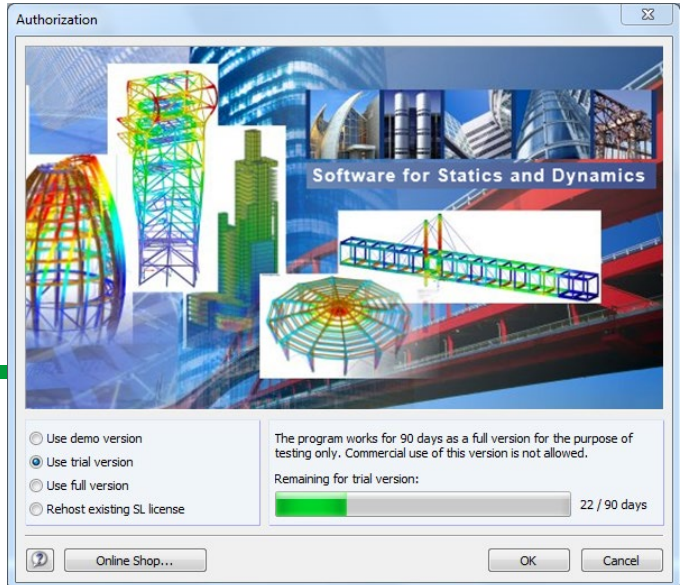
Help with Assignments

- Exercises / consulting hours
- Disscussion (MyCourses / Teams)
- Dlubal: https://www.dlubal.com/en



Additional Information

- Importer: Rak Tek Solutions Oy: http://www.rakteksolutions.fi/
- Free student licence, which is valid for one year: https://www.dlubal.com/en/education/students/free-structural-analysis-software-for-students
- First steps with RFEM (Manuals, tutorials and so on): https://www.dlubal.com/en/products/rfem-fea-software/first-steps-with-rfem



Backup!

The file is in C-disk.

Get a copy!

- MyCourses
- Email
- USB
- WIN-home folder
- etc.

Version history:

- Beam.rf5 (model being edit)
- Beam_v5.rf5
- Beam_v4.rf5
- Beam_v3.rf5
- Beam_v2.rf5
- Beam_v1.rf5

Rejoice in learning!