RFEM 5
Powerful, Easy, and Intuitive

Enjoy Structural Analysis

3D Finite Element Analysis
Software for All Industries

- Concrete structures
- Steel structures
- Timber structures
- Dynamic analysis
- Mechanical engineering
- Industrial engineering
- Piping systems
- Bridges
- Cranes and craneways
- Towers and masts
- Power plants
- Glass structures
- Tensile membrane structures
- Laminate structures, etc.
RFEM - Powerful & Intuitive
3D FEA Software

Universally Applicable
- Analysis of structures made of steel, concrete, timber, aluminium, glass, or combined structures
- Current national and many international standards
- Structural and dynamic analysis
- Linear and nonlinear analysis

Useful Generation Tools
- Structures including loads (e.g. continuous beams, frames, 3D halls)
- Generation of wind and snow loads according to Eurocode
- Parametrization of frequently recurring systems

Intuitive Graphical User Interface
- Quick induction into the software
- Creating structural and load data within a very short time
- Graphical or tabular model entry
- High-quality visualisation of the structure and loads
Dynamic Analysis

- Analysis of natural frequencies
- Seismic design
- Forced vibrations
- Time history method, multi-modal response spectrum method
- Nonlinear time history analysis

Modelling Supporting Structures

- Member, surface, and solid elements
- Support, member, and release nonlinearities
- Couplings and eccentricities for members and surfaces
- Nonlinear material models (plastic, thermal-elastic, etc.)

Form-Finding and Generation of Cutting Patterns for Membrane Structures

Connection Design

- Multilingual Printout Report
  - 13 languages available (independent of the input language)
  - Manual or automatic printout (mass print) of graphics
  - Export to RTF or PDF files
  - Possibility to create global printout templates
Numerous Member and Surface Types

- Beam
- Rigid
- Rib
- Truss
- Tension
- Compression
- Buckling
- Cable
- Cable on Pulleys
- Result Beam
- Definable Stiffness
- Coupling Rigid-Rigid etc.

Automatic Generation of Combinations According to Selected Combination Expressions

- Combinations according to Eurocode and many other international standards
- Consideration of accidental design situations (such as North German Plain)
- Options to reduce the number of generated combinations
- Copying, adding, etc. load cases in a well-arranged dialog box

Coloured Result Display on Rendered Model

- Freely adjustable range of values
- Animation of deformations, surface stresses and internal forces

Building Information Modeling (BIM)

- Direct interfaces with Tekla Structures, Autodesk Revit, and Bentley ISM
- Other exchange formats: DXF, IFC, STP (e.g. for Intergraph, Advance Steel, SEMA, cadwork, hsbcad), and DSTV (e.g. for Bocad and Frilo)
**About RFEM**

The 3D FEM structural analysis software RFEM is the basis of a modular software system. The main program RFEM is used to define structures, materials, and loads for planar and spatial structural systems consisting of plates, walls, shells and members. The program also allows you to create combined structures as well as model solid and contact elements.

RFEM provides deformations, internal forces, stresses, support forces, and soil contact stresses. The structure and load generators integrated in the program facilitate data input. Diverse add-on modules allow you to analyse plate and beam structures as well as connections according to various international standards.

The modular software concept allows you to compile a program package tailored to your individual needs. It is possible to upgrade the program at any time.
Enjoy Structural Analysis with Dlubal Software

In addition to operating comfort of the programs, Dlubal Software is known for its competent and fast support.

Furthermore, Dlubal Software website provides numerous information that can help you with your daily work: knowledge base, webinars, videos, manuals, FAQ, and many others.