



DLUBAL RFEM 5 ESSENTIAL TRAINING – STEEL STRUCTURES

Objectives:

After completing this training, you will be able to:

- ✓ Create a 3D structural model
- ✓ Define loads and load combination
- ✓ Calculate the internal forces, deformations and support reactions
- ✓ Analyse the member and surfaces stresses
- ✓ Design steel structures according to Standard Code
- ✓ Display results using graphic window and table
- ✓ Checking results according to Standard Code
- ✓ Optimize Cross-Section
- ✓ Generate printout report
- ✓ Import/Export AutoCAD with Dlubal RFEM

Modules:

RFEM 5 Main Module

Add on Modules:

RF-STEEL Members – General stress analysis of members

RF-STEEL Surfaces – General stress analysis of surfaces

RF-STEEL EC3 – Member and surface design according to Eurocode 3



TRAINING PROGRAMME DAY 1

Chapter 1: Introduction to Dlubal RFEM 5

- 1.1 Finite Elements
- 1.2 Graphical User Interface
- 1.3 Project Management
- 1.4 Units and Decimal Places
- 1.5 Type of Model 2D/3D
- 1.6 Standard Code

PART 1: MODEL, ANALYZE AND DESIGN STEEL MEMBERS.

Chapter 2: Entering the Model Data

- 2.1 Creating a New Project
- 2.2 Creating a New Model
- 2.3 Moving, Zooming, Rotating
- 2.4 Adjust Work Window and Grid
- 2.5 Defining Members
- 2.6 Creating Members
- 2.7 Placing the Horizontal Beam
- 2.8 Showing Numbering
- 2.9 Dividing a Member
- 2.10 Defining Tapered Members
- 2.11 Connecting Members Eccentrically
- 2.12 Placing Ceiling Joist as Continuous
Members
- 2.13 Connecting Members
- 2.14 Defining Nodal Supports
- 2.15 Defining Member End Releases

- 2.16 Changing the Numbering

- 2.17 Checking the Input

Chapter 3: Assigning Load

- 3.1 Load Case 1: Self-Weight
- 3.2 Load Case 2: Snow
- 3.3 Load Case 3: Wind lateral on Columns
- 3.4 Load Case 4: Wind Lifting
- 3.5 Load Case 5 to 7: Imposed Load
- 3.6 Load Case 8: Imperfections

Chapter 4: Combination of Actions

- 4.1 Defining Load Combinations
- 4.2 Defining Result Combinations

Chapter 5: Calculation

- 5.1 Checking Input Data
- 5.2 Starting Calculation

Chapter 6: Results

- 6.1 Available Results
- 6.2 Deformations, Internal Forces, Support Forces
- 6.3 Results Display
- 6.4 Result Diagrams on Members
- 6.5 Multiple Windows View
- 6.6 Filter Results
- 6.7 Animation of Deformations



TRAINING PROGRAMME DAY 2

Add-on Modules: RF-STEEL Members

Chapter 7: Input Data

- 7.1 General Data
- 7.2 Materials
- 7.3 Cross-Sections

Chapter 8: Calculations

- 8.1 Detail Settings
- 8.2 Stresses and Ratio
- 8.3 Start Calculation

Chapter 9: Results

- 9.1 Stresses by Cross-Section, Set of Members, Member, x-Location, at Every Stress Point
- 9.2 Governing Internal Forces by set of Members
- 9.3 Part List by Member and Set of Members
- 9.4 Cross Section Optimization

Chapter 10: Results Evaluation

- 10.1 Selection of Stresses
- 10.2 Results on Cross-section
- 10.3 Results in RSTAB Model
- 10.4 Result Diagrams
- 10.5 Filter for Results

Add-on Modules: RF-STEEL EC3

Chapter 10: Input Data

- 10.1 ULS, SLS, Fire Resistance, National Annex
- 10.2 Materials
- 10.3 Cross-Sections
- 10.4 Intermediate Lateral Restraints
- 10.5 Effective Lengths
- 10.6 Nodal Supports
- 10.7 Member Hinges
- 10.8 Serviceability Data
- 10.9 Fire Resistance
- 10.10 Parameters

Chapter 11: Calculation

- 11.1 Detail Settings
- 11.2 Ultimate Limit State
- 11.3 Stability
- 11.4 Serviceability
- 11.5 Fire Resistance
- 11.6 Other
- 11.7 Start Calculation



TRAINING PROGRAMME DAY 2

Chapter 12: Results

- 12.1 Design by Load Case, Cross-Section, Set of Members, Member, x-location
- 12.2 Governing Internal Forces by set of Members and Set of Members
- 12.3 Member Slenderness
- 12.4 Parts List by Member and Set of Members
- 12.5 Cross Section Optimization

Chapter 13: Results Evaluation

- 13.1 Results on RSTAB Model
- 13.2 Result Diagrams
- 13.3 Filter Results

Chapter 14: Printout

- 14.1 Printout Report
- 14.2 Graphic Printout

PART 2: MODEL, ANALYZE AND DESIGN STEEL SURFACES.

Chapter 15: Entering the Model Data

- 15.1 Settings
- 15.2 Base Plate
- 15.3 Column
- 15.4 Manual Definition of Surfaces
- 15.5 Generation of Surfaces from Member
- 15.6 Rigid End Plate and Supports

Chapter 16: Load Data

- 16.1 Load Case 1: Self-Weight
- 16.2 Load Case 2: Imposed Load
- 16.3 Superimposing the Loads

Chapter 17: FE Mesh Settings

- 17.1 Node Refinements
- 17.2 Line Refinements

Chapter 18: Calculation & Results

- 18.1 Plausibility Check
- 18.2 Starting Calculation
- 18.3 Results

Add-on Modules: RF-STEEL Surfaces

Chapter 19: Data and Results

- 19.1 Input Data
- 19.2 Results
- 19.3 Printouts

Chapter 20: General Functions

- 20.1 Design Cases
- 20.2 Import/Export of Materials
- 20.3 Units and Decimal Places
- 20.4 Export Results
- 20.5 Data Transfer AutoCAD-RFEM