



Structural Analysis & Design Software

www.dlubal.com



Dipl.-Ing. (FH) Andreas Hörold
Organizer

Marketing & Public Relations
Dlubal Software GmbH



Ing. Doğukan Karataş, M.Sc.
Co-Organizer

Technical Product Manager
Dlubal Software GmbH



Ing. Jaroslav Brož, PhD
Co-Organizer

Scrum Master
Dlubal Software Prague

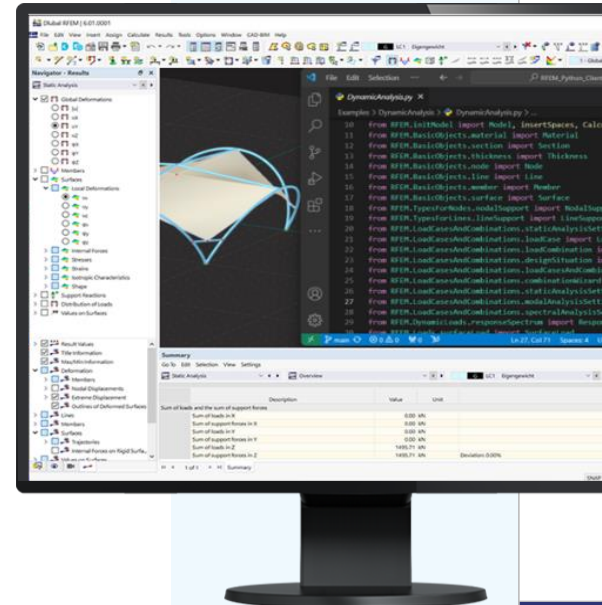


Dipl.-Ing. Rebecca Göbel
Co-Organizer

Product Engineering & Customer Support
Dlubal Software GmbH

Webinar

Advanced Structural Analysis with RFEM 6 Python API



Questions During the Presentation



GoToWebinar Control Panel
Desktop



E-mail: info@dlubal.com



Show or hide
control panel



Adjust audio
settings



Ask questions

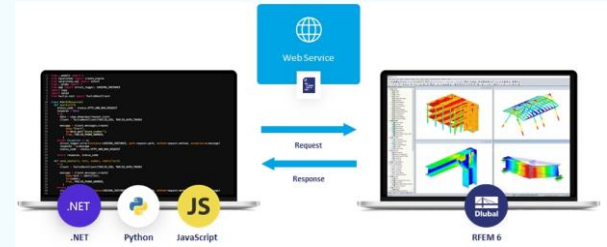


The screenshot shows the GoToWebinar control panel interface. At the top, there is a menu bar with 'File', 'View', and 'Help'. Below it is the 'Audio' section, which includes a 'Sound Check' indicator with a green bar and a question mark. There are two radio buttons: 'Computer audio' (selected) and 'Phone call'. A red 'MUTED' indicator is visible. Below this, there are dropdown menus for 'Mikrofon (2- Sennheiser USB h...)' and 'Lautsprecher (2- Sennheiser U...'. A volume slider is also present. The 'Questions' section is below the audio settings, featuring a text input field with the placeholder '[Enter a question for staff]' and a 'Send' button. At the bottom, the 'Webinar ID: 373-901-987' and the 'GoToWebinar' logo are displayed.



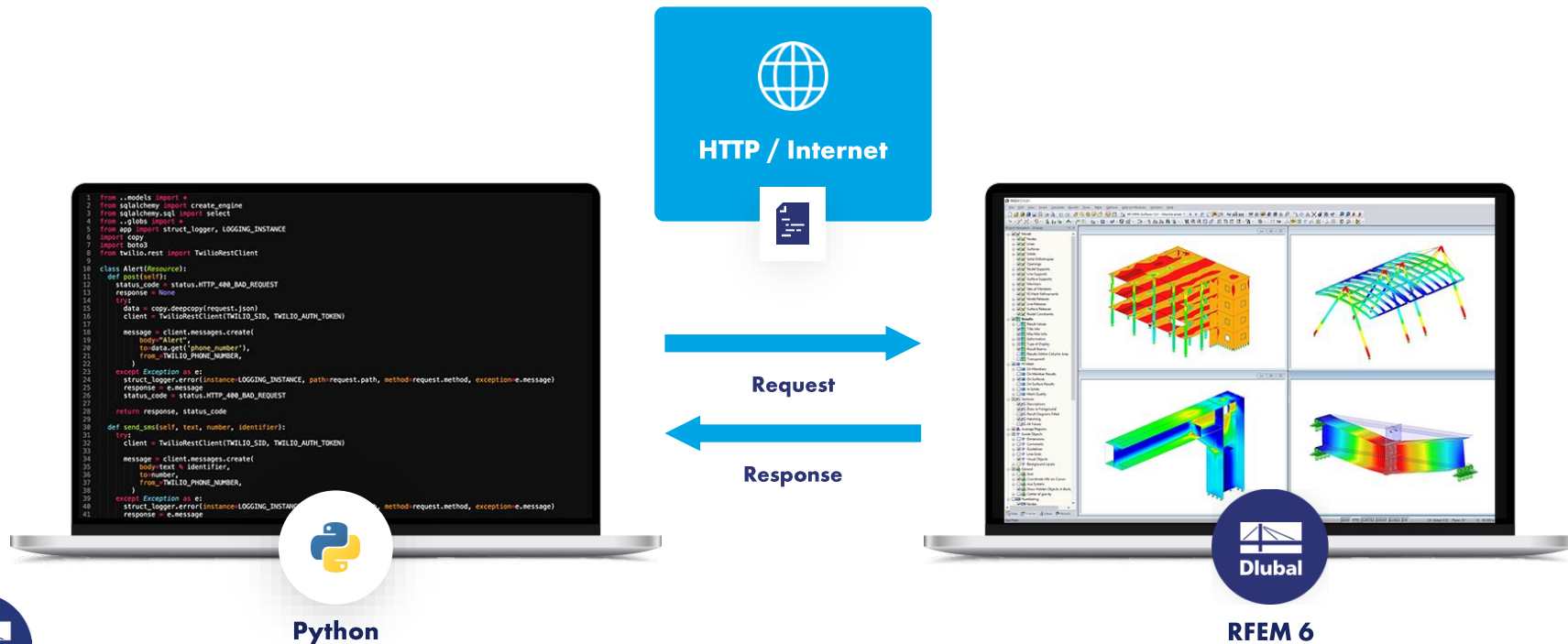
CONTENT

- 01 Overview of Webservices & API
- 02 Latest Implementation to Python Library
- 03 Library and Community
- 04 Examples





What is API? What are Webservices?





Recent Development

Calculate in Cloud

```
Examples > CalculateInCloudExample > CalculateInCloudExample.py > ...
39 Model.clientModel.service.finish_modification()
40
41 saveFile(dirName + "\Cloudtest.rf6")
42
43 # Define on which server the cloud calculation should be run
44 server_name = "Dlu_1"
45
46 # No plausibility check should be done before cloud calculation is started
47 run_plausibility_check = False
48
49 # When False cloud calculation is not started when there are errors in plausibility check
50 calculate_despite_warnings_and_errors = False
51
52 # When true, email notifications for start and end of cloud calculation are sent
53 email_notification = True
54
55 result = CalculateInCloud(server_name, run_plausibility_check, calculate_despite_warnings_and_errors)
56
57 # First list object = task ID of cloud calculation task
58 # Second list object = status of calculation
59 if result:
60     task_id = result[0]
61     status = result[1]
62     print(f"Task-ID: {task_id}")
63     print(f"Status: {status}")
```

```
dirname: c:\Users\KaratasD\Desktop\pyDev\RFEM_Python_Client\Exam
ples\CalculateInCloudExample
Connecting to server...
Cloud calculation was started.
Task-ID: 4734
Status: Uploaded and queued
PS C:\Users\KaratasD\Desktop\pyDev\RFEM_Python_Client>
```



Recent Development

Mesh Tables

Navigator - Data

LC1 - Self-weight
Loads (kN)

steelBeamSurfaces.rft*

- Basic Objects
 - Materials
 - Sections
 - Thicknesses
 - Nodes
 - Lines
 - Members
 - Surfaces
 - Openings
 - Solids
 - Line Sets
 - Member Sets
 - Surface Sets
 - Solid Sets
- Special Objects
- Types for Nodes
- Types for Lines
- Types for Members
- Types for Surfaces
- Types for Solids
- Types for Special Objects
- Imperfections
- Load Cases & Combinations
 - Load Cases
 - Design Situations
 - Load Combinations
 - Static Analysis Settings
 - Load Wizards
 - Loads
 - LC1 - Self-weight
 - Calculation Diagrams
 - Results
 - Guide Objects
 - Printout Reports

Finite Element Nodes

Go To Edit Selection View Settings

Finite Element Mesh

FE Node No.	X [m]	Y [m]	Z [m]
1	0.000	0.000	0.000
2	0.000	0.000	0.350
3	0.000	-0.100	0.000
4	0.000	0.100	0.000
5	0.000	-0.100	0.350
6	0.000	0.100	0.350
7	2.000	0.000	0.000
8	2.000	0.000	0.150
9	2.000	-0.100	0.000
10	2.000	0.100	0.000
11	2.000	-0.100	0.150

H 1 of 2 Nodes 2D Elements

Navigator - Results

Static Analysis

- Global Deformations
 - ux
 - uy
 - uz
 - qx
 - qy
 - qz
- Surfaces
 - Layer Side
 - Local Deformations
 - Internal Forces
 - Stresses
 - Strains
 - Isotropic Characteristics
- Shape
- Support Reactions
- Distribution of Loads
- Values on Surfaces

LC1 - Self-weight
Loads (kN)
Static Analysis
Displacements [μ (mm)]

max |u| : 901.1 | min |u| : 0.0 mm

Surfaces | Local Deformations

Go To Edit Selection View Settings

Static Analysis Results by Surface

Surface No.	Grid Point No.	X	Y	Z	u	ux	uy
1	2	0.500	0.000	0.000	75.3	37.1	-65.5
	3	1.000	0.000	0.000	250.5	65.3	-241.8
	4	1.500	0.000	0.000	531.0	87.4	-523.6
	5	2.000	0.000	0.000	896.1	97.1	-892.7
	Extremes	5	2.000	0.000	0.000	896.1	97.1
1	1	0.000	0.000	0.000	0.0	0.0	0.0
	5	2.000	-0.000	0.000	896.1	97.1	-892.7
	1	0.000	0.000	0.000	0.0	0.0	0.0
	5	2.000	0.000	0.000	896.1	97.1	-892.7
	5	2.000	0.000	0.000	896.1	97.1	-892.7

H 2 of 19 Global Deformations Local Deformations Basic Internal Forces Principal Internal Forces Design Internal Forces



Recent Development

MIA Assistant

Structural Analysis and Design Software

Efficient Wind Load Simulation Using CFD for All Structure Types

The CFD program RWIND 2 revolutionizes the determination of wind loads for building structures. It only needs a 3D model of your project and a wind load description; all other parameters for the flow simulation are determined automatically.

The wind pressure and suction are then seamlessly integrated into the structural analysis program RFEM or RSTAB for a comprehensive structural analysis.

Discover the precision and efficiency of RWIND 2 for your wind load simulations. Get started now and take your projects to new levels utilizing the most advanced CFD technology.

[MORE ABOUT CFD PROGRAM RWIND 2](#)

FULL VERSION FOR TESTING

Hi Mia! How can I activate my Add-Ons in RFEM6?

To activate Add-Ons in RFEM 6, follow these steps:

1. Open the 'Edit Model - General Data' dialog in RFEM 6.
2. Go to the 'Add-Ons' tab within the dialog.
3. Check the box next to the Add-On you wish to activate.
4. The activated Add-On will then be displayed in the Navigator.

If you need further assistance with activating Add-Ons or have specific questions about the process, feel free to ask.

Where did this answer come from?

Hi, I'm Mia, your AI assistant. Ask

Shift + Enter to add new line

Stahbauanschluss...

Enjoy Structural Analysis & Design Increasing Performance with Cloud Revolution in Wind Load Analysis: RFEM 6: Complete Solution for Your Seismic Stability by Pushover



Recent Development

MIA Assistant

The screenshot displays the RFEM 6 software interface. The main window shows a 3D perspective view of a cube with a coordinate system. The left sidebar contains a 'Navigator - Data' tree with categories like Basic Objects, Materials, Settings, Thicknesses, Nodes, Lines, Members, Surfaces, Openings, Solids, Line Sets, Member Sets, Surface Sets, Solid Sets, Special Objects, Types for Nodes, Types for Lines, Types for Members, Types for Surfaces, Types for Solids, Types for Special Objects, Imperfections, Load Cases & Combinations, Actions, Design Stations, Action Combinations, Load Combinations, Static Analysis Settings, and Combination Wizards. The bottom panel shows a 'Materials' table with the following data:

Material No.	Material Name	Material Type	Material Model	Modulus of Elast. E [N/mm ²]	Shear Modulus G [N/mm ²]	Poisson's Ratio ν [-]	Specific Weight γ [N/m ³]	Mass Density ρ [kg/m ³]	Coeff. of Th. Exp. α [1/°C]	Options	Comment
1	Steel	Steel	Isotropic Linear Elastic	210000.0	80769.2	0.300	78.50	7850.00	0.000012		
2											
3											
4											
5											
6											
7											
8											
9											
10											

The right sidebar features a chat window titled 'Your AI Assistant MIA'. The chat contains the following text:

Hi MIA! How can create a modal analysis load case in RFEM6?

To create a modal analysis load case in RFEM 6, please follow these steps:

1. Open the 'Load Cases and Combinations' dialog in the RFEM 6 interface.
2. Navigate to the 'Load Cases' tab within the dialog.
3. Click the 'New Load Case' button to create a new load case.
4. In the 'Edit Load Case' dialog, select 'Modal Analysis' as the 'Type of Analysis'.
5. Define the necessary parameters for the modal analysis, such as the number of eigenvalues, the method for determining them, and any other specific settings required for your analysis.
6. If needed, you can also define which masses should be considered in the modal analysis by selecting the relevant load cases or combinations that contain the masses.
7. Confirm your entries by clicking 'OK' to create the modal analysis load case.

After setting up the load case, you can proceed with the calculation, and upon completion, RFEM 6 will provide you with the results, including:

Hi, I'm MIA, your AI assistant. Ask anything in your language... (Shift + Enter to add new line)



Library and Community

The screenshot shows the GitHub Discussions interface for the repository 'Dlubal-Software / RFEM_Python_Client'. The page is dark-themed and features a navigation bar at the top with options like Code, Issues, Pull requests, Discussions (selected), Actions, Projects, Wiki, Security, Insights, and Settings. A search bar is located in the top right corner.

The main content area is divided into two columns. The left column contains 'Categories' (General, Ideas, Polls, Q&A, Show and tell) and a 'Most helpful' section listing 'heetrojivadiya' as the top contributor. The right column displays a list of discussions, each with a title, author, date, and status. The discussions are sorted by 'Latest activity' and filtered to show 'Open' discussions.

Discussion Title	Author	Date	Status	Reactions
How to get member internal forces by member	nathanwanthof	asked on Jun 27, 2023 in Q&A	Unanswered	3
How to generate response spectrum with standards ?	icarelbnFirmas	asked on Nov 6, 2023 in Q&A	Unanswered	1
Internal forces default axes	cyclonehopper	asked last week in Q&A	Unanswered	0
Status Update on Printout Report Feature	Jeroen124	asked on Feb 21 in Q&A	Unanswered	0
Optimizing Data Extraction from RFEM Models	Jeroen124	asked on Feb 21 in Q&A	Unanswered	0
Setting material stiffness	felipebandeirani	asked on Dec 23, 2022 in Q&A	Answered	5
Connect to remote PC	MaximilianFranz	asked on Oct 13, 2023 in Q&A	Unanswered	11
Steel Design	MaartenBW	asked on Sep 18, 2023 in Q&A	Unanswered	1



Library and Community

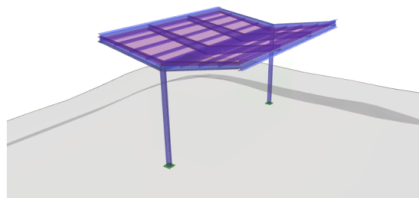
 Dlubal Dev Docs

[Home](#)

[User Guide](#)

[About](#)

[Github](#)



RFEM 6 / RSTAB 9 | Webservices & API

Do Cool Stuff in Structural Analysis

[Get Started](#) 

High Level Library

Program easy and smart by using the power of RFEM 6 / RSTAB 9 beyond their graphical interfaces.

Open Source

Contribute to the public development to see the source code and learn from others.

Optimize & Parametrize

Create your own apps for fast and smooth workflows.





Book your free Online Appointment!

Get valuable insights from one of our experts



**Dipl.-Ing. (FH) Dipl.-
Wirtschaftsing. (FH)
Christian Stautner**

Head of Sales



Bastian Ackermann, M.Sc.

Sales



Daniel Dlubal, M.Sc.

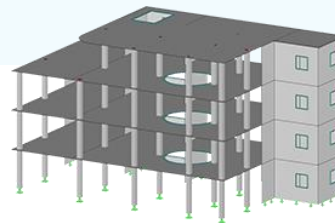
COO of Dlubal Software GmbH



 **Contact Our Sales Team**

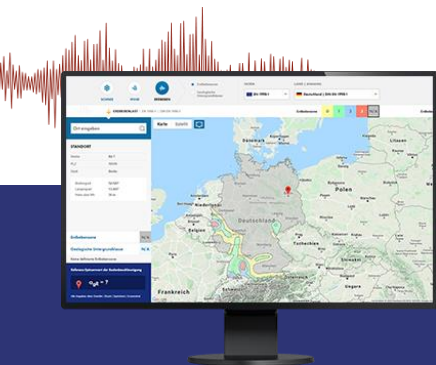


Free Online Services



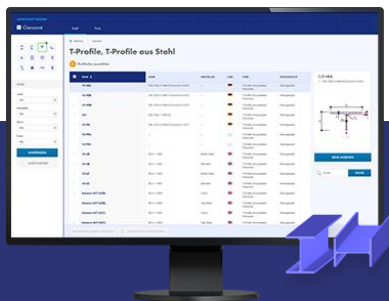
Geo-Zone Tool

Dlubal Software provides an online tool with snow, wind and seismic zone maps.



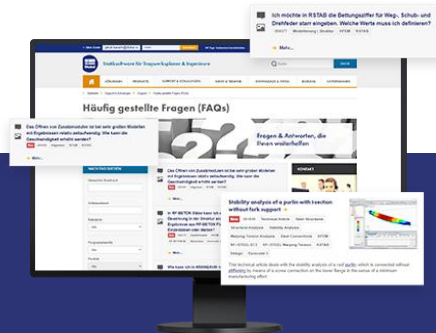
Cross-Section Properties

With this free online tool, you can select standardized sections from an extensive section library, define parametrized cross-sections and calculate its cross-section properties.



FAQs & Knowledge Base

Access frequently asked questions commonly submitted to our customer support team and view helpful tips and tricks articles to improve your work.



Models to Download

Download numerous example files here that will help you to get started and become familiar with the Dlubal programs.





Free Online Services

Youtube Channel - Webinars, Videos

Videos and webinars about the structural engineering software.



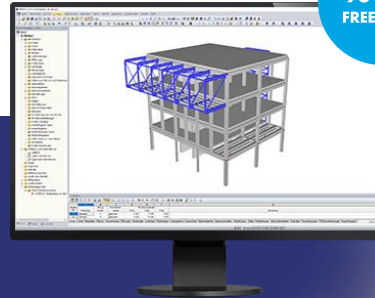
Webshop with Prices

Configure your individual program package and get all prices online!



Trial Licenses

The best way how to learn using our programs is to simply test them for yourself. Download a 90-day free trial version of our structural analysis & design software.



90-DAY FREE TRIAL



We offer free support via email and chat



— Get Further Details About Dlubal



Visit website

www.dlubal.com

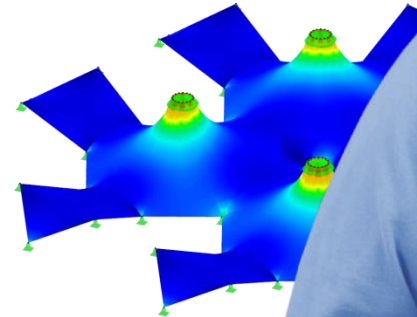
- Videos and recorded webinars
- Newsletters
- Events and conferences
- Knowledge Base articles



See Dlubal Software in action in a webinar



Download free trial license



Dlubal Software GmbH
Am Zellweg 2,
93464 Tiefenbach, Germany

Phone: +49 9673 9203-0
E-mail: info@dlubal.com



www.dlubal.com