![STADATA STRUCTURAL SOFTWARE logo]

3Muri Project Training Hub:
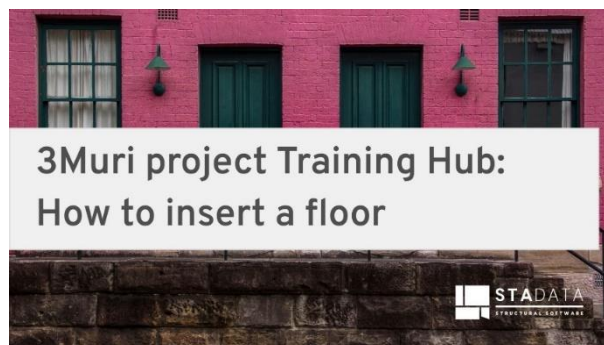
How to insert a floor

Abstract

S.T.A. DATA walks viewers through the process of adding a floor in the 3Muri Project structural modeling software.

Introduction

S.T.A. DATA introduces Training Hub, a list of specialized videos on 3Muri Project structural modelling.

This video guides the professional technician on the process of a floor modeling in 3Muri. Adding a floor in a structural model is a crucial step for creating a structure, especially for seismic analysis. To start, click the floor icon and choose the floor type from a predefined list or saved configurations. Then, enter the floor's parameters, such as thickness, materials, and geometry. Define the floor's position by selecting reference structural elements and setting the warping direction.


After defining the loads acting on the floor, adjust other parameters like support length, safety coefficients, and color/texture. Finally, visualize the load distribution to ensure accuracy before proceeding with the analysis.

How to insert the floor in 3Muri Project

Next steps show the procedure to add a floor in 3Muri Project model.

1) Click on the floor icon on the toolbar in the structure environment.

This icon will bring
up the floor
insertion tool
where we'll do
most of the work.
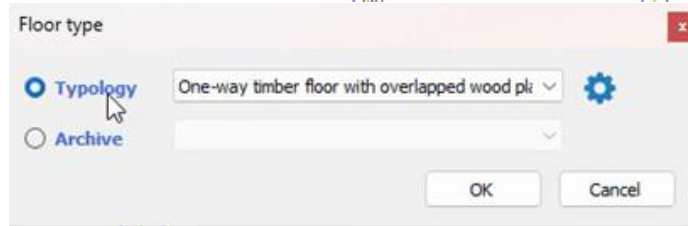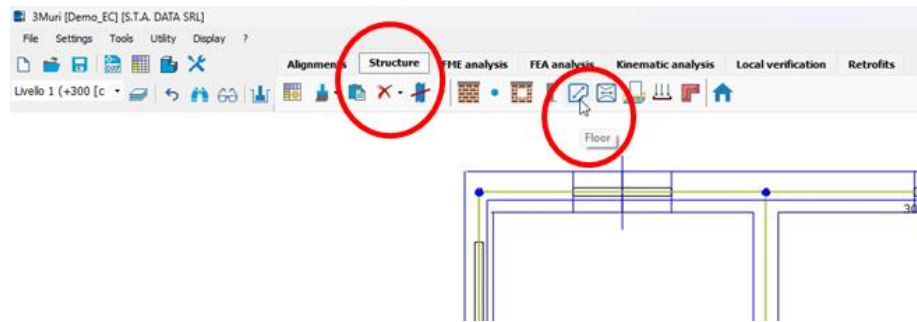
2) Select the floor type.

A predefined typology list
includes options like wooden
floors, concrete slabs or
composits. If you have saved floor
configuration you can also load
one from the archive.
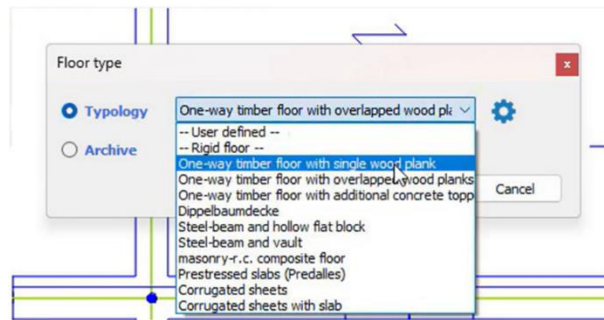
3) Input floor parameters.
The key parameters starts by defining thickness that impact the floor's behavior in the
model. After these parameters we'll input the materials for the floor.
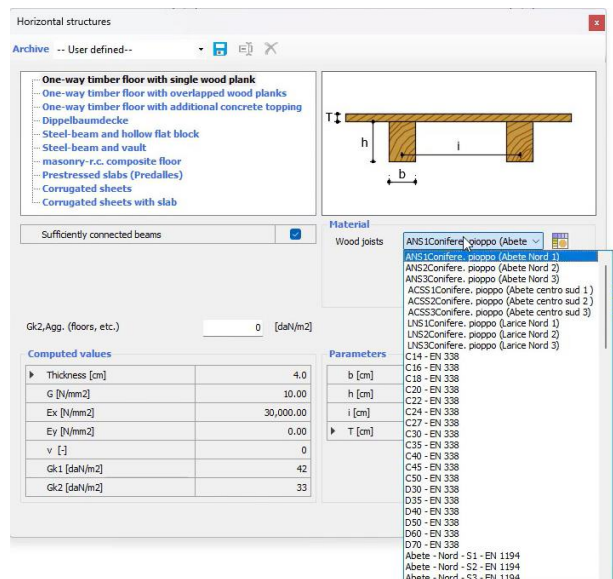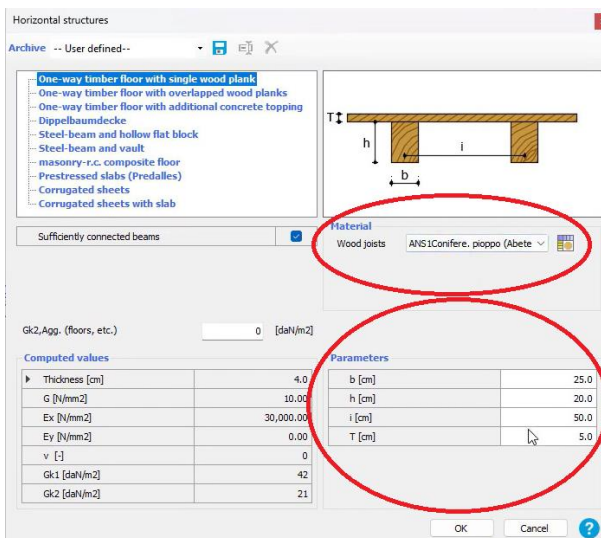
4) Set the geometry of the floor.

This step involves selecting the points that make up the boundaries of the floor.

5) Select the reference structural elements and warping the direction.

The reference structural elements support the floor like beam and walls. The direction of warping defines how loads transfer across the floor and into the supporting elements.
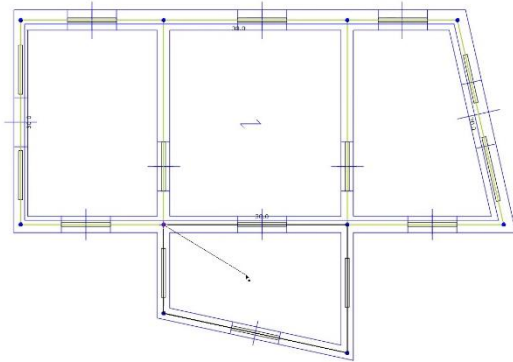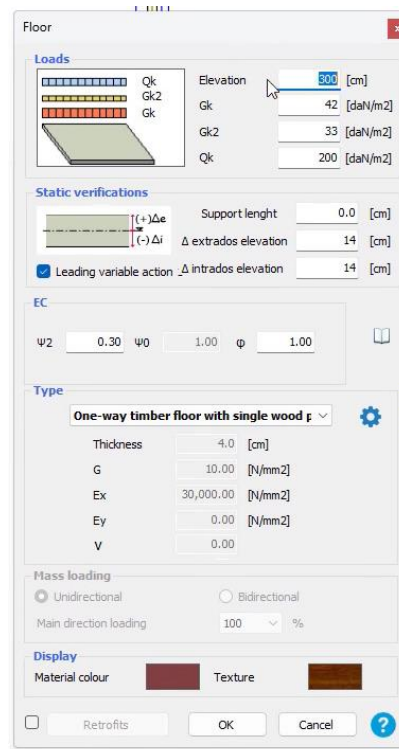
6) Set loads acting on the floor.

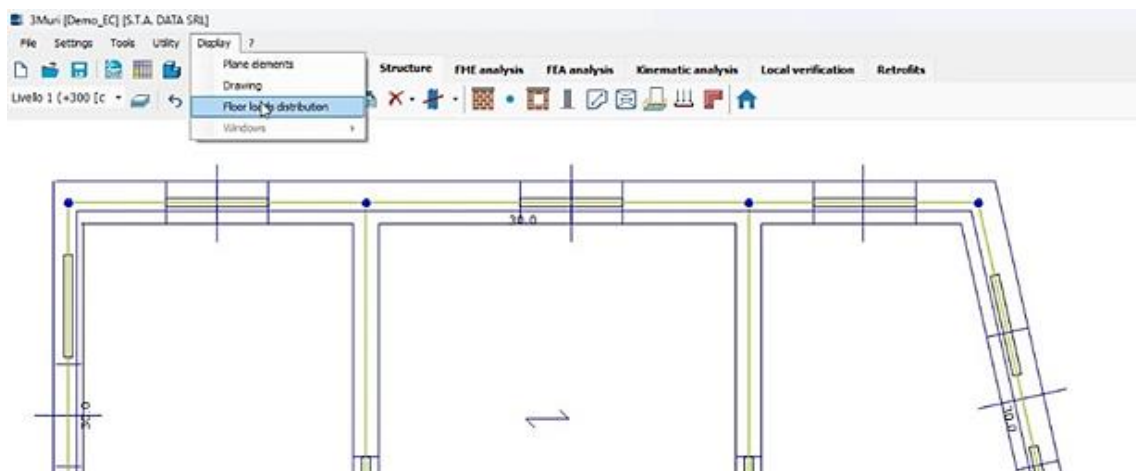Now it's time to define loads acting on the floor.

7) Set other parameters.

You can adjust additional parameters. Define support length for the floor's edges, input safety coefficient to meet local standards.
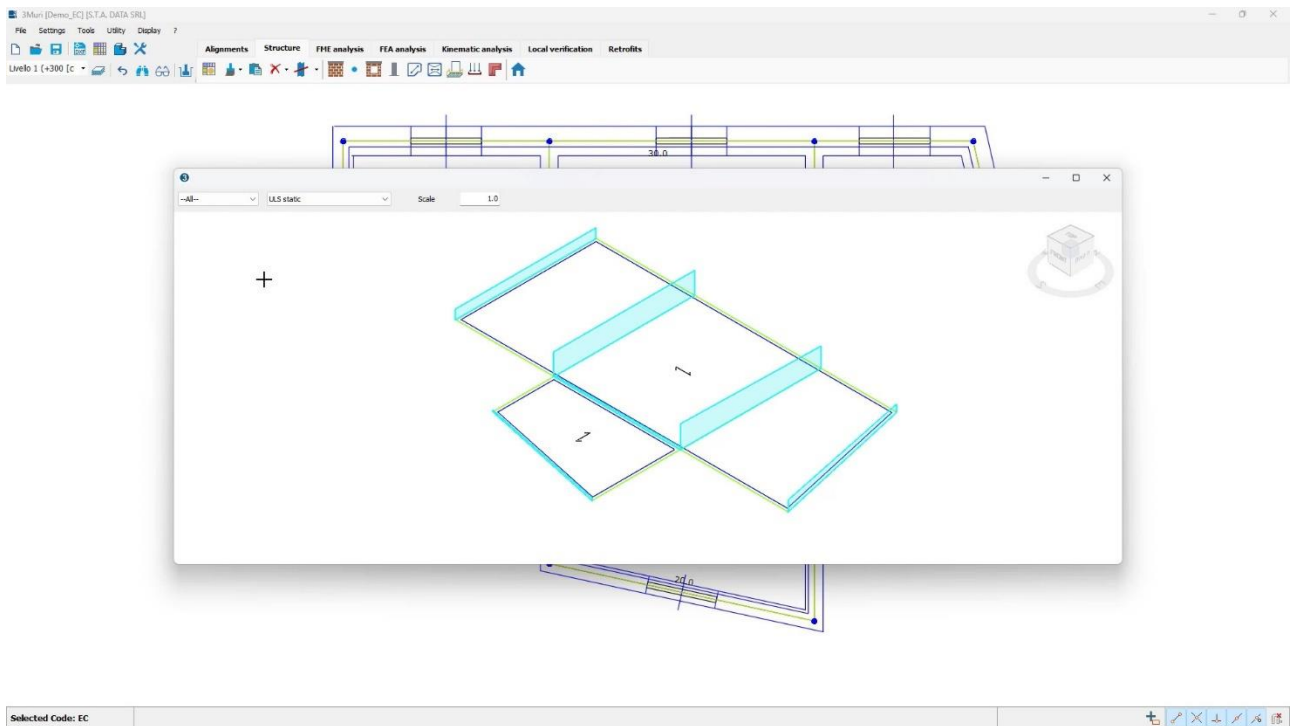

8) Visualize load distribution.

To make sure everything is good to go, head over Display icon and select the floor load distribution.

This option will show exactly how the load are distributed across your floor.

9) Ready for analysis.

Your floor is fully configured and ready to run the analysis. You can move on checking how it performs under the given conditions.