Shell Element Formulation

Overview

FEAKalc 3D employs a three-dimensional finite element shell formulation for the analysis of slabs, walls, plates and other surface structures. Shell elements combine membrane behaviour, plate bending and transverse shear deformation within a single element, allowing structures to resist both in-plane and out-of-plane actions.

Each shell node possesses six degrees of freedom comprising three translations and three rotations. Consequently, shell elements connect directly to beam, column and support elements without requiring special interface elements, allowing complete three-dimensional structural models to be analysed within a common global stiffness system.

The shell formulation has been developed specifically for FEAKalc 3D and is based on established finite element principles used throughout structural engineering.

Quadrilateral Shell Element

The primary shell element used by FEAKalc 3D is a four-node isoparametric quadrilateral element. The element combines three independent stiffness contributions:

-membrane stiffness
-plate bending stiffness
-transverse shear stiffness

The membrane formulation is based on conventional bilinear plane-stress interpolation.

Plate bending follows the Mindlin-Reissner thick plate theory, allowing transverse shear deformation to be represented in addition to classical bending behaviour. This makes the element suitable for both thin and moderately thick structural components.

To improve transverse shear behaviour, FEAKalc 3D employs an assumed-natural-strain (ANS) style interpolation based on edge tying points. Rather than computing transverse shear directly from the standard bilinear displacement field, the shear strains are reconstructed from covariant shear quantities evaluated at specific locations along the element edges. This significantly reduces the tendency for shear locking commonly associated with low-order Mindlin shell elements.

The quadrilateral formulation therefore represents a custom Mindlin shell incorporating ANS/MITC4-inspired transverse shear treatment together with explicit shear softening and drilling-rotation stabilisation.

Membrane and bending stiffness are integrated using a conventional 2×2 Gauss integration scheme.

Triangular Shell Element

For irregular geometries, FEAKalc 3D also provides a three-node triangular shell element.

The triangular element combines:
-constant strain membrane behaviour
-Mindlin plate bending
-softened transverse shear formulation
-drilling rotation stabilisation

Unlike the quadrilateral formulation, transverse shear is evaluated using a centroid-based formulation with additional shear softening.

The triangular element provides excellent geometric flexibility and is particularly useful for irregular boundaries, openings and automatic mesh generation. However, owing to its different interpolation functions and transverse shear formulation, it should generally be regarded as a transition or closure element rather than the preferred element for large regular slab or wall regions.

Drilling Rotation Stabilisation

Classical shell theory does not provide stiffness about the shell normal (drilling rotation). Without additional stabilisation this degree of freedom produces a singular element stiffness matrix.

FEAKalc 3D introduces an artificial drilling stiffness about the local shell normal to stabilise this rotational degree of freedom. The drilling stiffness does not represent a physical structural stiffness but improves numerical stability and conditioning of the global stiffness matrix.

Material Behaviour

The present shell formulation assumes linear elastic isotropic material behaviour.
Membrane stiffness follows conventional plane stress elasticity while bending stiffness varies with the cube of shell thickness in accordance with classical plate theory.

Local Coordinate System

Each shell element possesses its own local coordinate system.
The local 1 and 2 axes lie within the shell plane while the local 3 axis is normal to the shell surface.
Element stiffness is assembled in local coordinates before being transformed into the global structural coordinate system for assembly into the global stiffness matrix.
Internal membrane forces, bending moments and transverse shear forces are also recovered in the shell local coordinate system.The current implementation adopts a Poisson's ratio of 0.20 within the shell element formulation.

Meshing

FEAKalc 3D provides two shell meshing options.

Quadrilateral Mesh

The quadrilateral mesh uses four-node shell elements throughout the model wherever possible.

This is the preferred meshing option for:
-floor slabs
-raft foundations
-retaining walls
-shear walls
-bridge decks
-regular plate structures

Quadrilateral elements generally provide the most accurate representation of bending and transverse shear behaviour and are recommended for engineering design.

Triangular Mesh

The triangular mesh automatically subdivides shell regions into three-node shell elements.

Triangular meshes are particularly useful for:
-irregular boundaries
-curved geometries
-complex openings
-automatic mesh generation

Although the triangular formulation provides excellent geometric flexibility, it employs a different transverse shear formulation to the quadrilateral element.

Consequently, users should expect small differences in transverse shear distributions between triangular and quadrilateral meshes.

These differences are a characteristic of the underlying finite element formulations rather than an indication of modelling error.

Connectivity

During mesh generation FEAKalc 3D automatically establishes compatibility between adjoining shell regions and connected frame members.

Common edges are subdivided consistently, coincident nodes are merged into the global model and frame members are connected directly into the shell mesh.

This allows slabs, walls, beams and columns to participate within a single compatible finite element model.

Loading

Distributed surface loading is converted into equivalent nodal loads prior to global analysis.

These equivalent nodal forces are assembled into the global structural load vector together with frame, nodal and support loads.

Result Recovery

Following solution of the global displacement vector, shell element forces and stresses are recovered in local element coordinates.

FEAKalc 3D reports:

-membrane forces (F11, F22 and F12)
-bending moments (M11 and M22)
-transverse shear forces (V13 and V23)
-membrane stresses
-transverse shear stresses

Results are recovered at the element centre and reported in the shell local coordinate system.

Comparison with ETABS

The theoretical basis of the FEAKalc 3D shell formulation is broadly comparable to the thick-shell formulation available in ETABS.

Both programs employ shell elements that combine membrane action, plate bending and transverse shear deformation within a six-degree-of-freedom nodal formulation.

Both programs also utilise local shell coordinate systems for recovery of internal membrane forces, bending moments and transverse shear forces.

Although the underlying theory is similar, the implementations are independent.

ETABS employs proprietary shell technology developed by Computers and Structures, Inc. (CSI), whereas FEAKalc 3D uses an independently developed shell formulation incorporating ANS/MITC4-inspired transverse shear treatment, custom drilling-rotation stabilisation and its own meshing and result recovery procedures.

Consequently, exact element-by-element agreement should not be expected.

Meaningful verification is achieved by comparing:

-support reactions
-nodal displacements
-membrane force distributions
-bending moment distributions
-transverse shear distributions

using equivalent geometry, material properties, boundary conditions, shell thicknesses, mesh densities and loading.

As mesh refinement increases, both programs should demonstrate convergence towards comparable engineering results.

Verification

The shell formulation implemented in FEAKalc 3D has been verified through comparison with analytical benchmark problems and independent finite element solutions produced using ETABS.

Verification studies compare:

-support reactions
-maximum displacements
-membrane forces
-bending moments
-transverse shear forces

under identical modelling assumptions.

Minor local differences may occur near concentrated loads, supports, re-entrant corners and regions of stress concentration owing to differences in meshing strategy, numerical integration, result recovery and proprietary implementation details.

Verification therefore focuses on overall structural behaviour, equilibrium, displacement response and converged internal force distributions rather than exact agreement of individual element contour values.

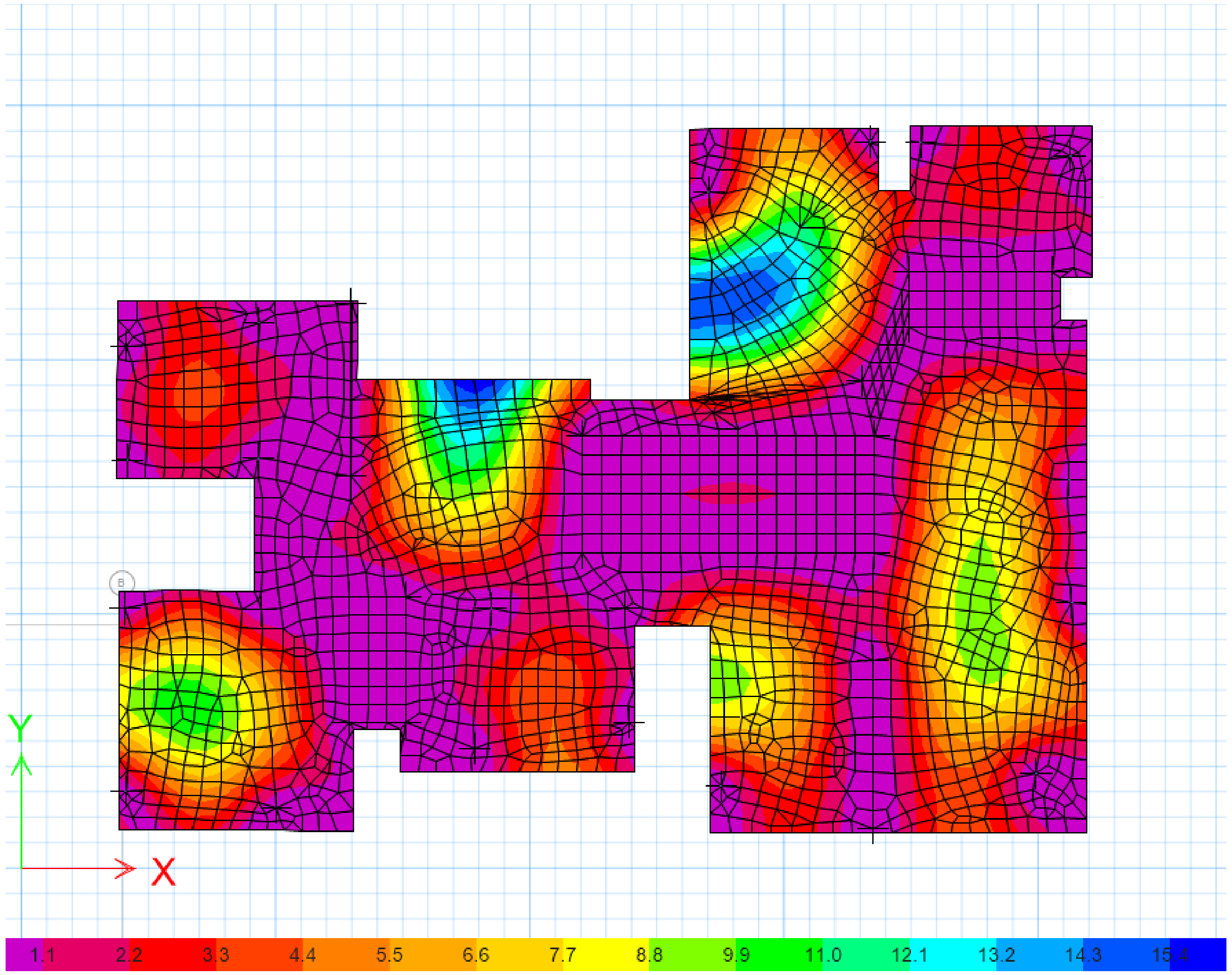
The comparisons demonstrate that FEAKalc 3D produces results consistent with established finite element practice while maintaining an independently developed shell formulation specifically tailored to the FEAKalc 3D analysis engine.

LEARNING CENTRE

Analysis Verification

Concrete Frame	By	RK
FEAKalc 3D Verifications	Sheet No. 7	Date 7/26

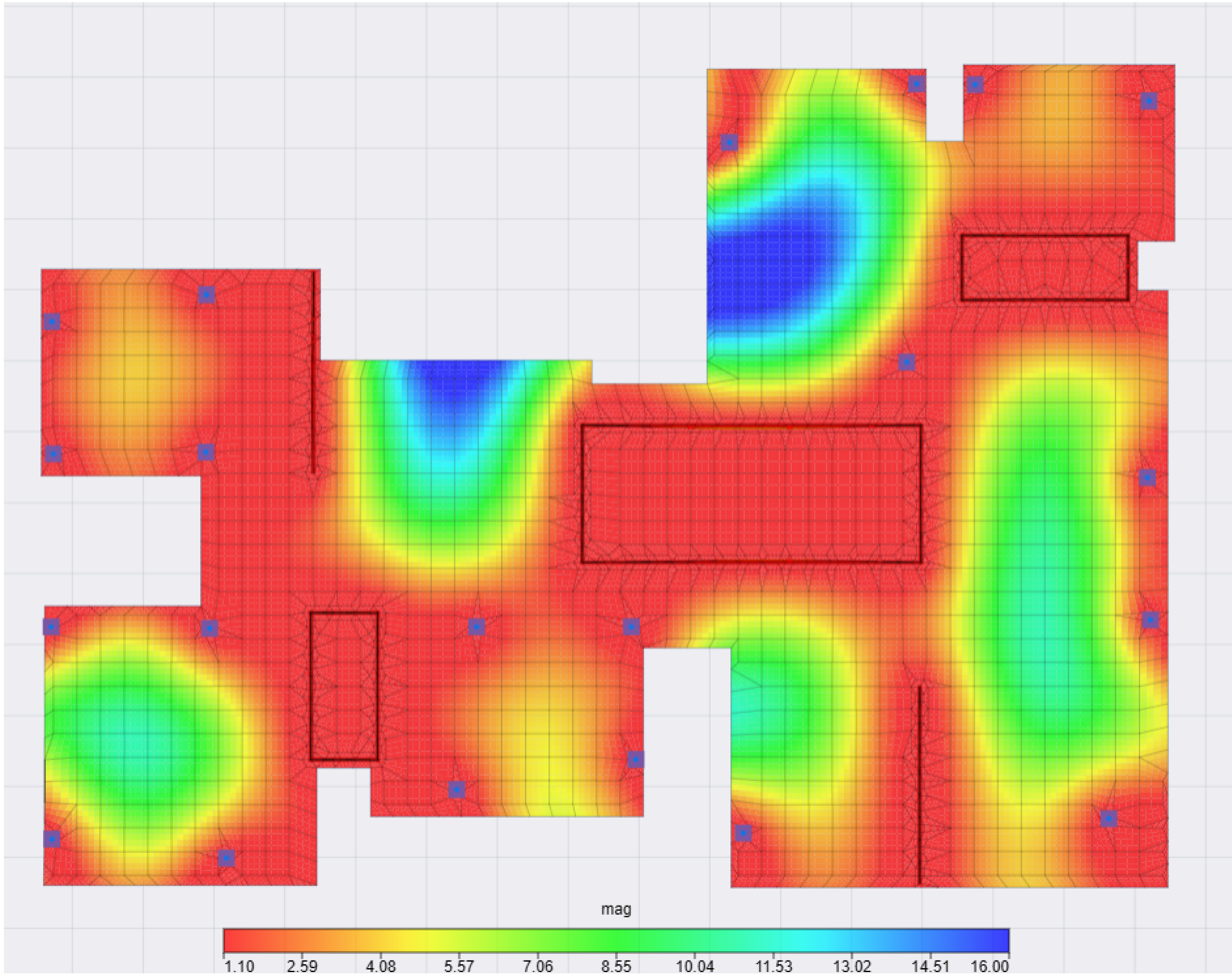
ETABS



Deformation Diagram

Under Self Weight

FEAKalc 3D

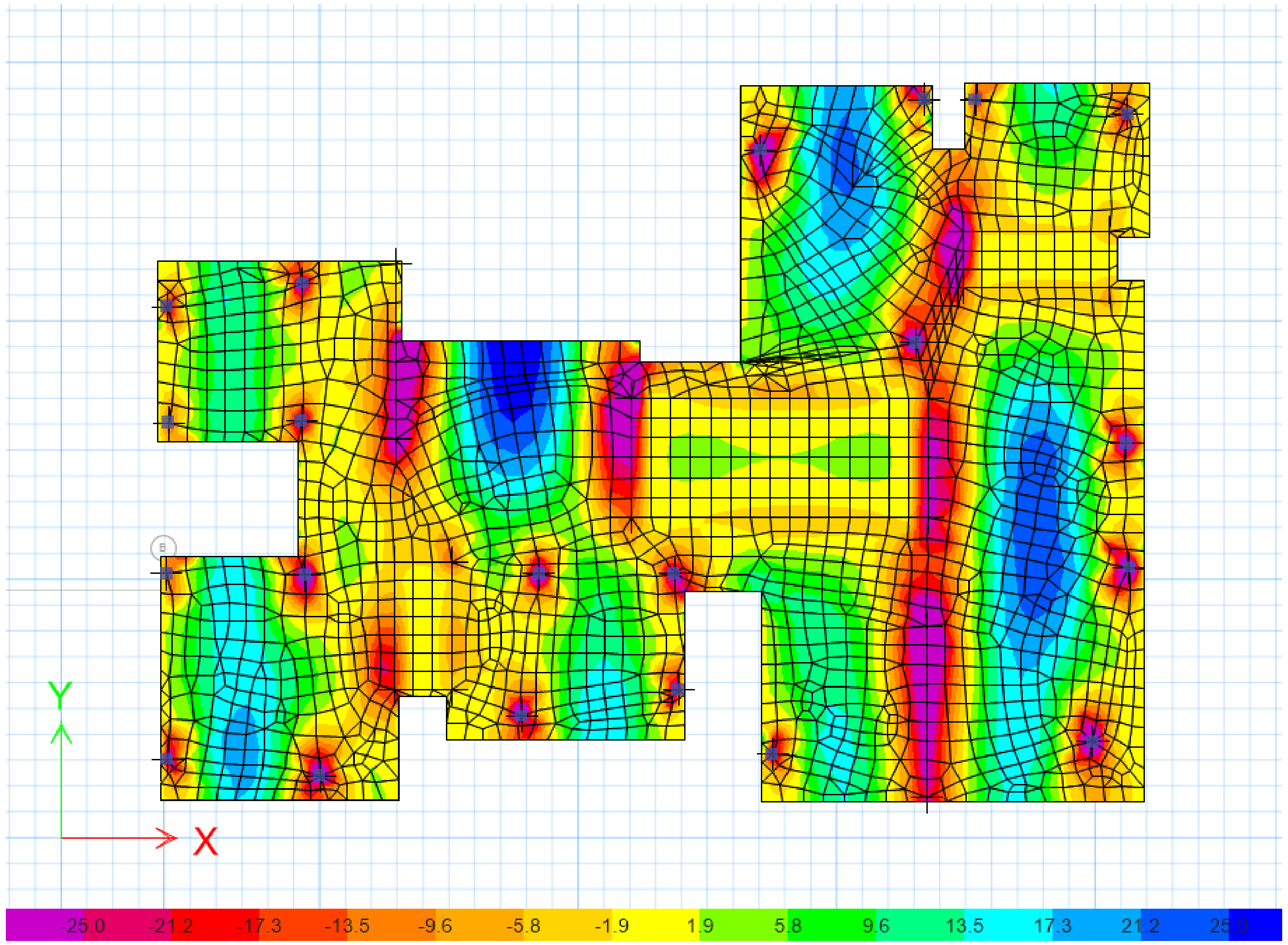


Deformation Diagram

Under Self Weight

RKALC LEARNING CENTRE	**Analysis Verification**	Concrete Frame	By	RK
		FEAKalc 3D Verifications	*Sheet No.* 8	*Date* 7/26

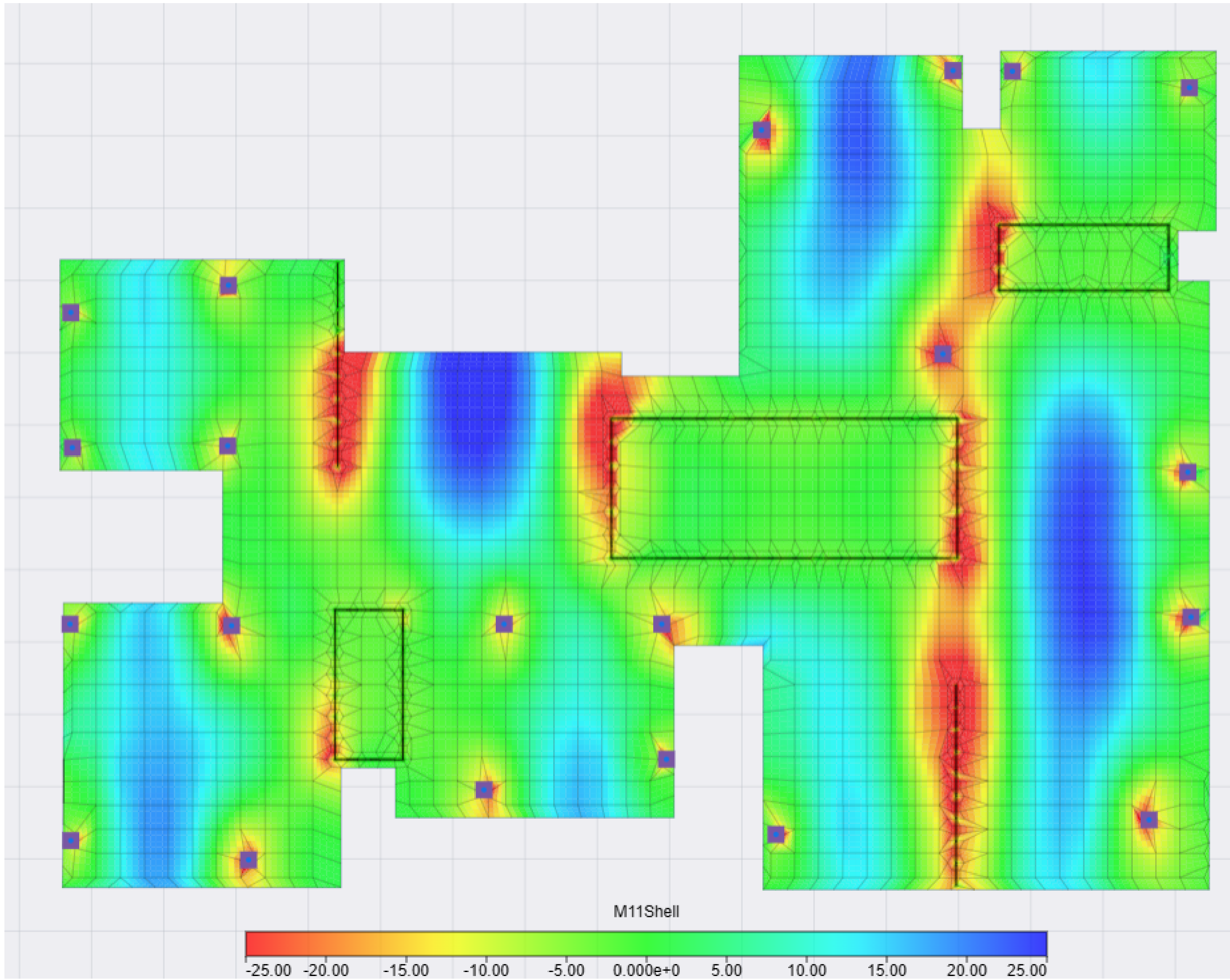
ETABS



M11 Shell Diagram

Under Self Weight

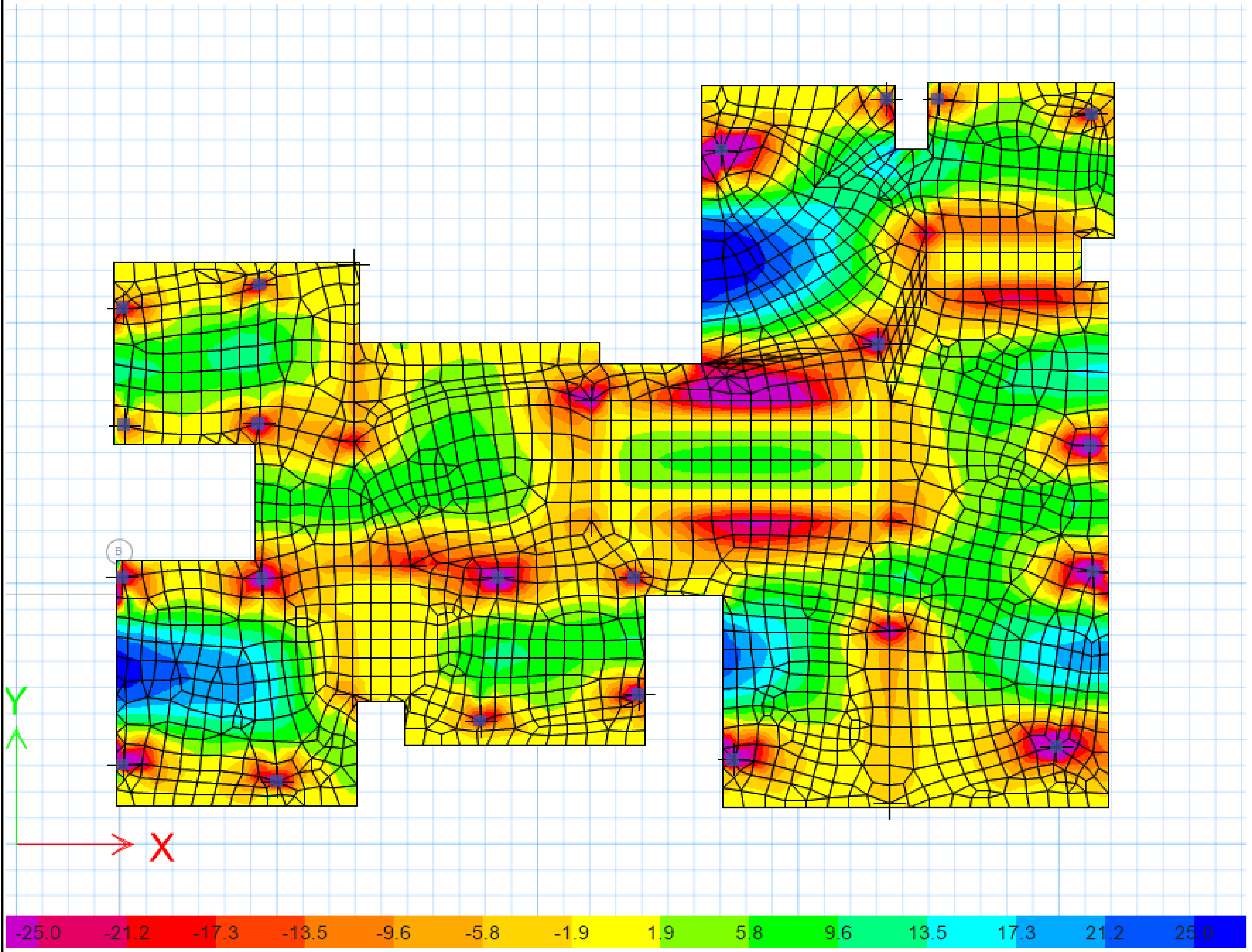
FEAKalc 3D



M11 Shell Diagram

Under Self Weight

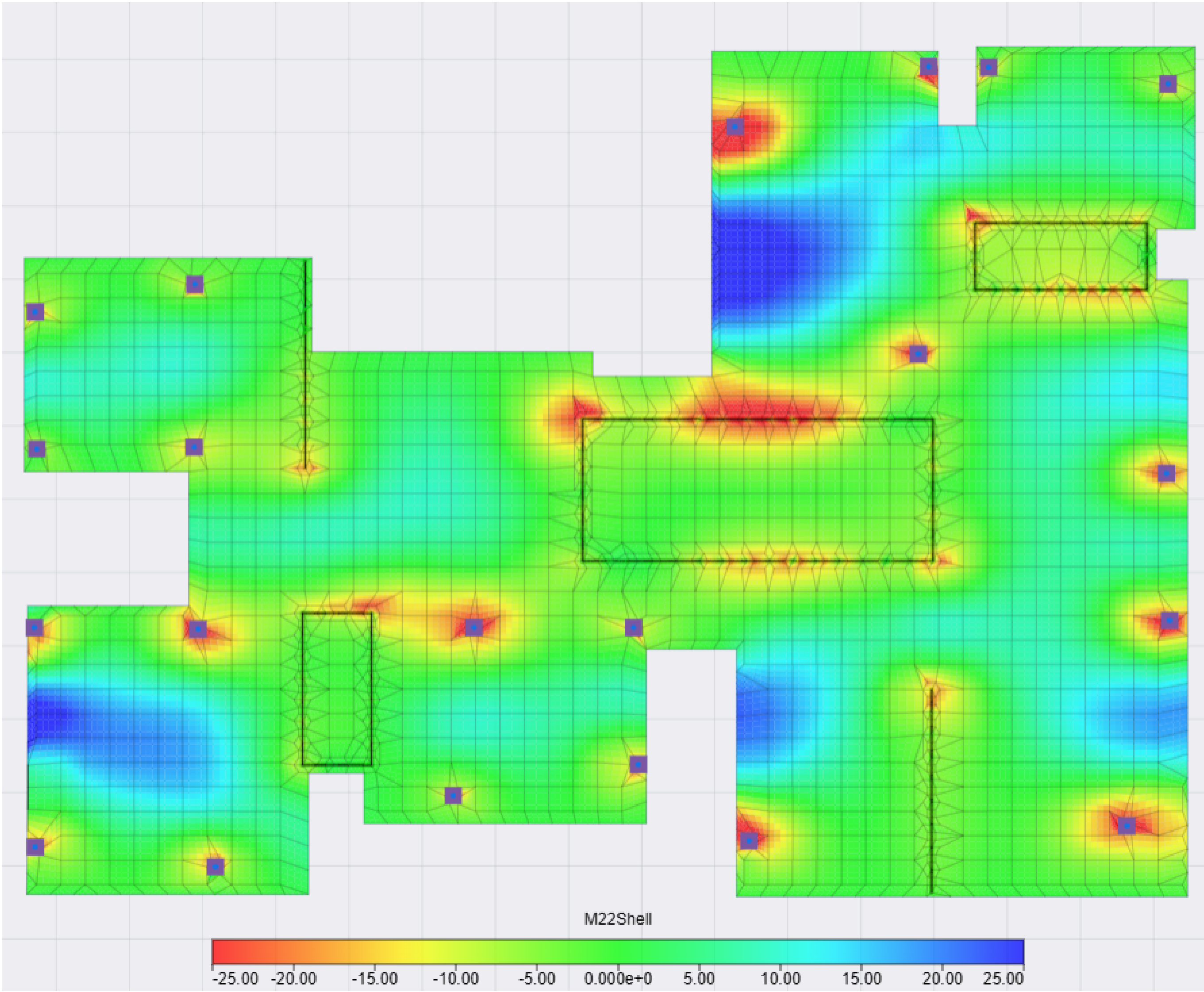
ETABS



M22 Shell Diagram

Under Self Weight

FEAKalc 3D



M22 Shell Diagram

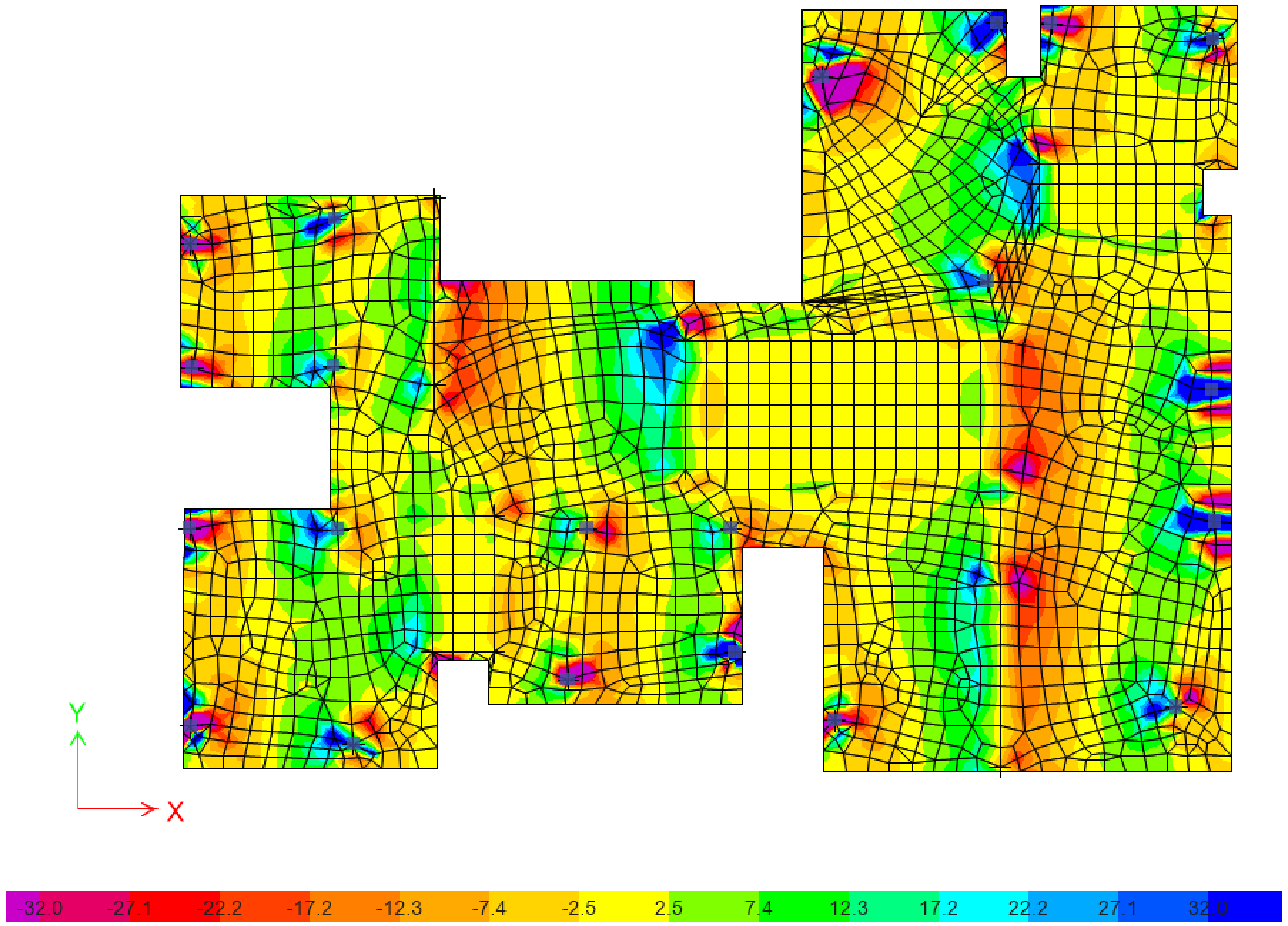
Under Self Weight

RKALC LEARNING CENTRE

Analysis Verification

Concrete Frame	By	RK
FEAKalc 3D Verifications	Sheet No. 10	Date 7/26

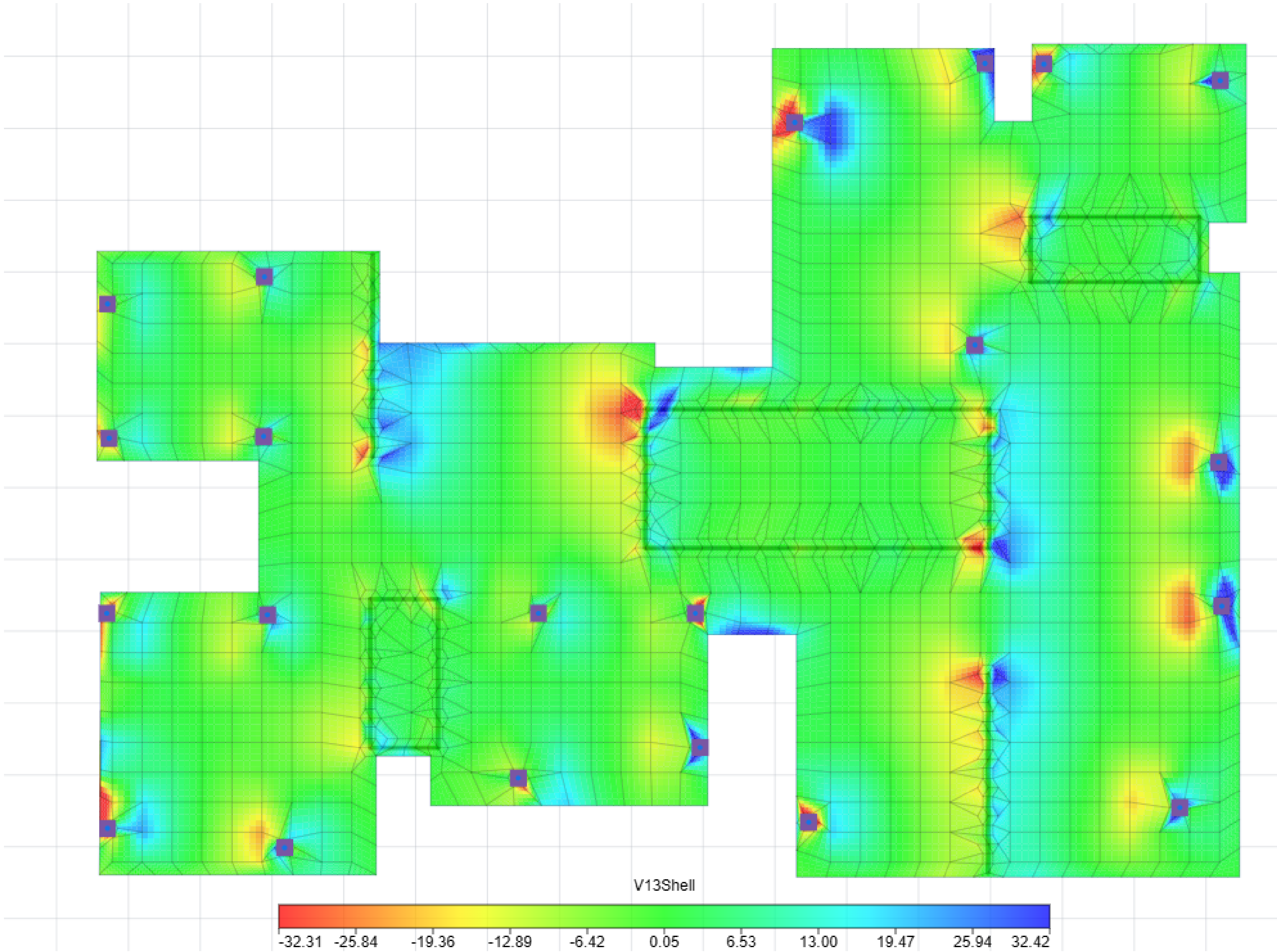
ETABS



V13 Shell Diagram

Under Self Weight

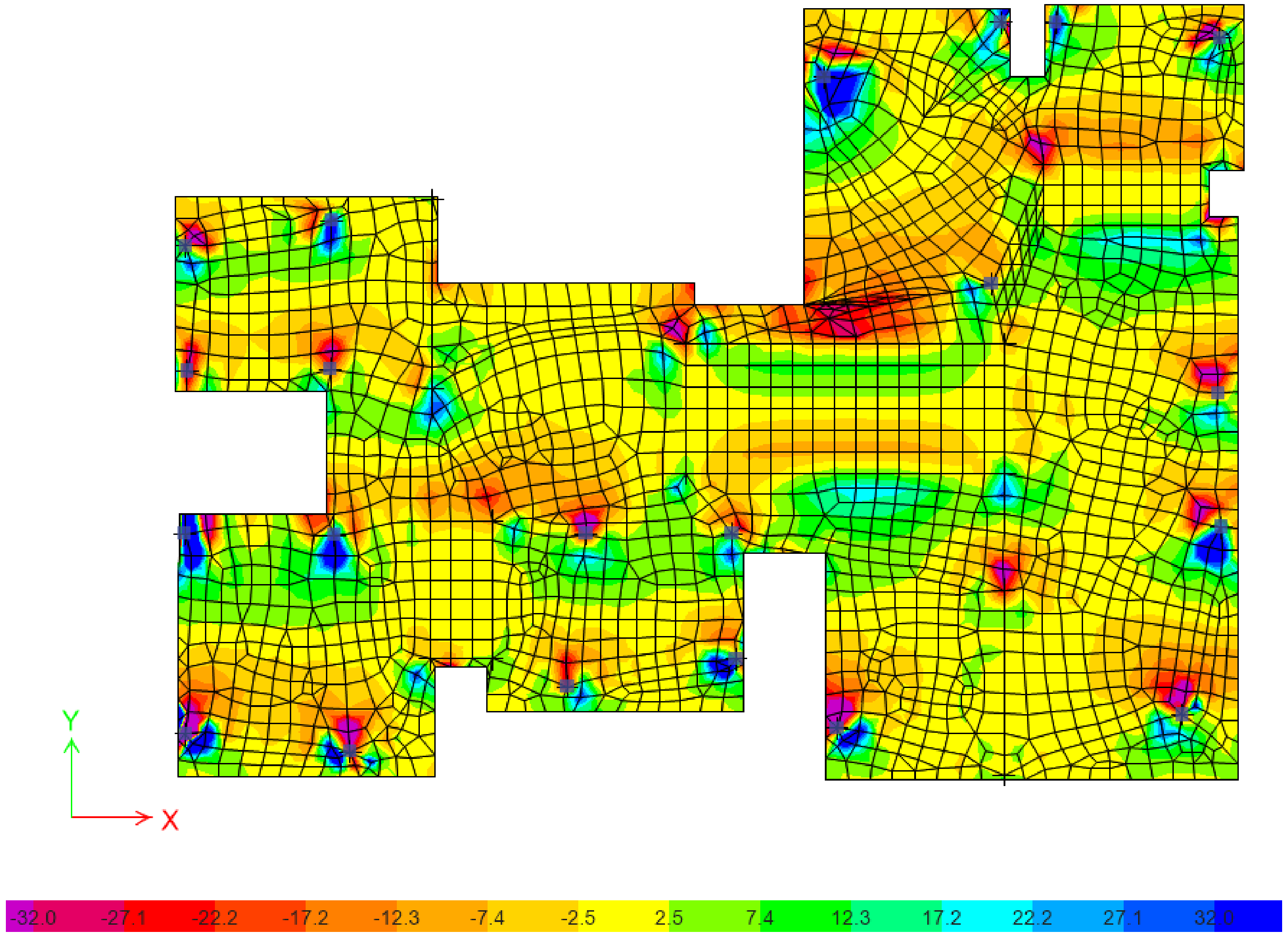
FEAKalc 3D



V13 Shell Diagram

Under Self Weight

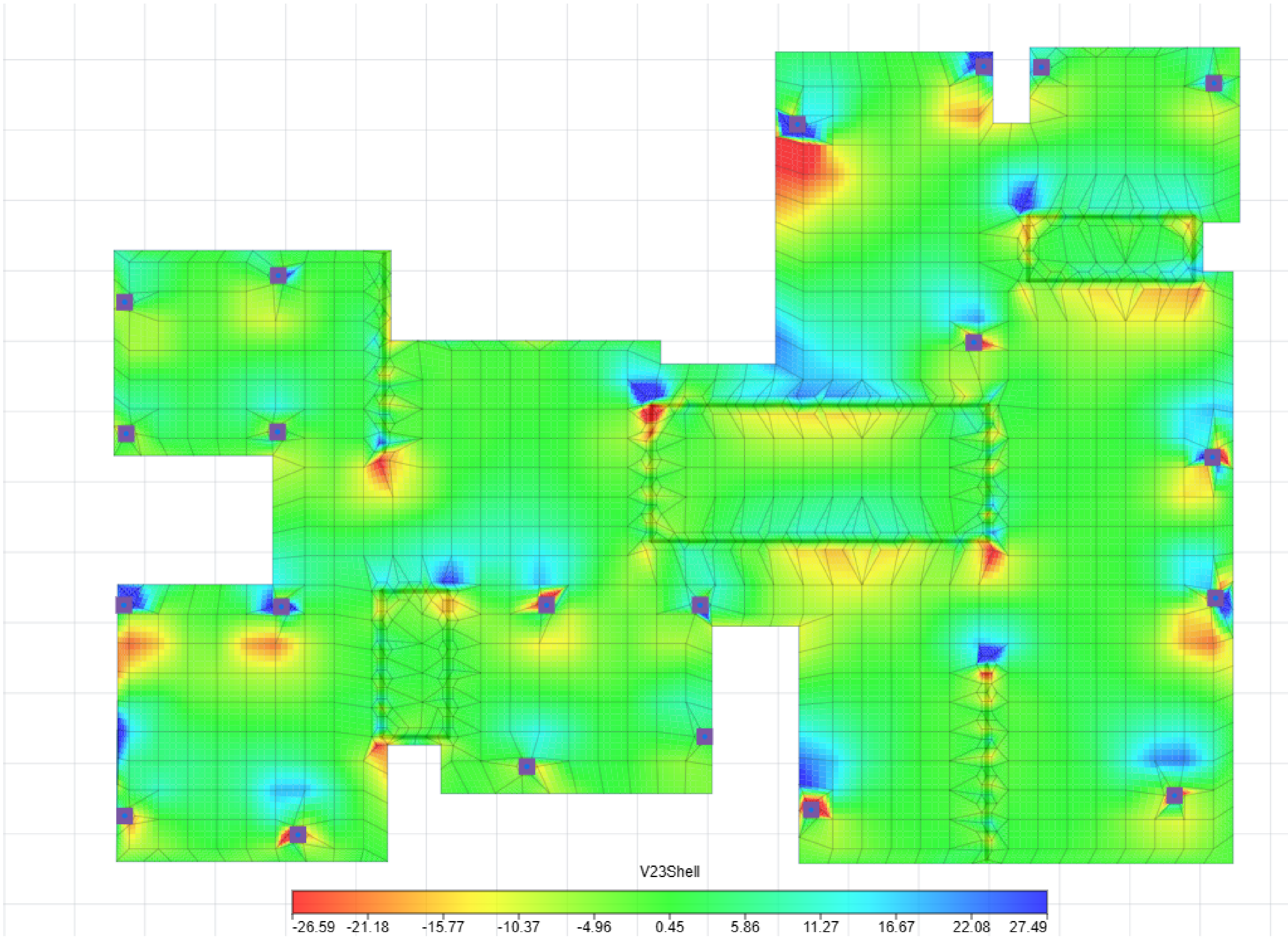
ETABS



V23 Shell Diagram

Under Self Weight

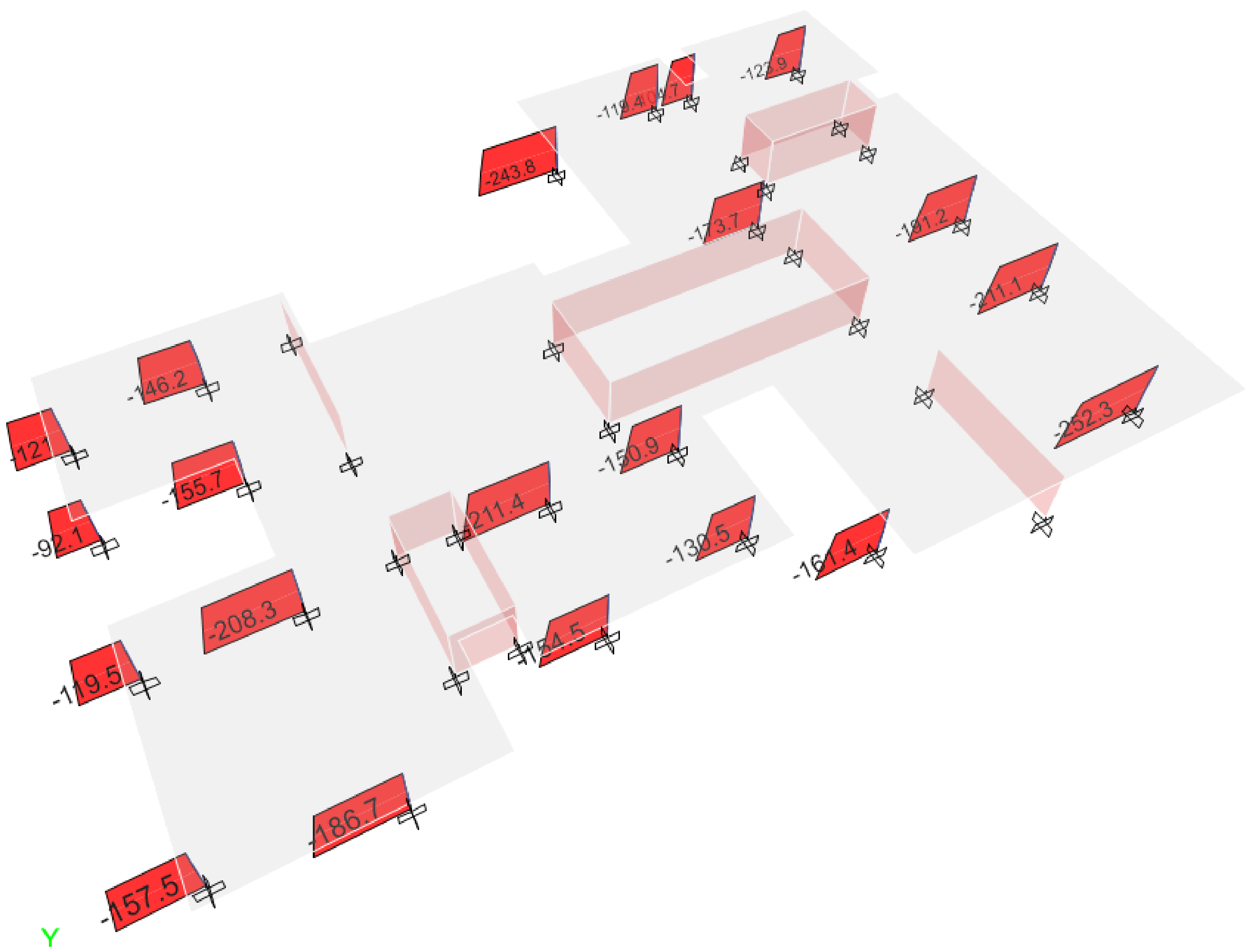
FEAKalc 3D



V23 Shell Diagram

Under Self Weight

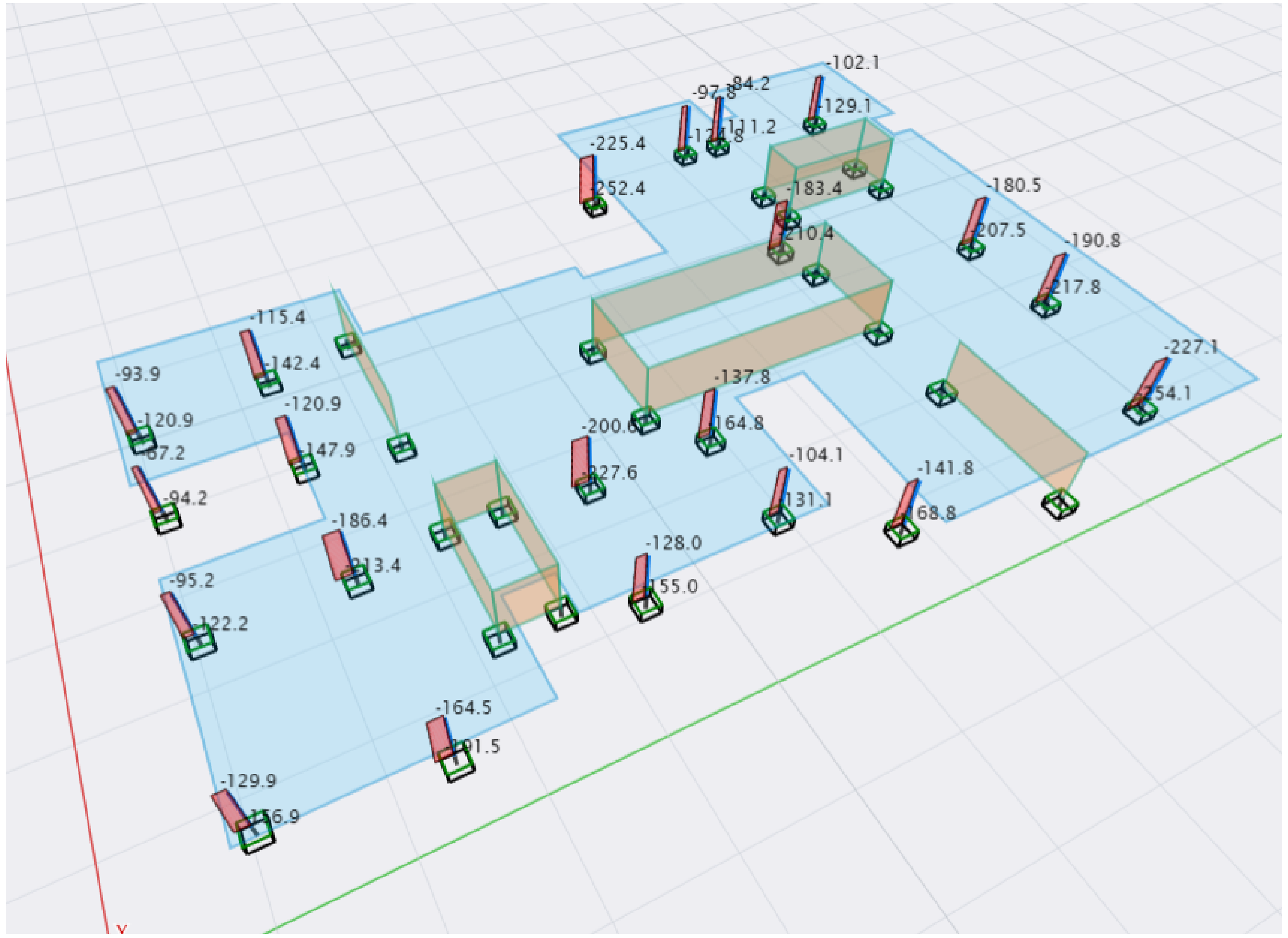
ETABS



Axial Force Diagram

Under Self Weight

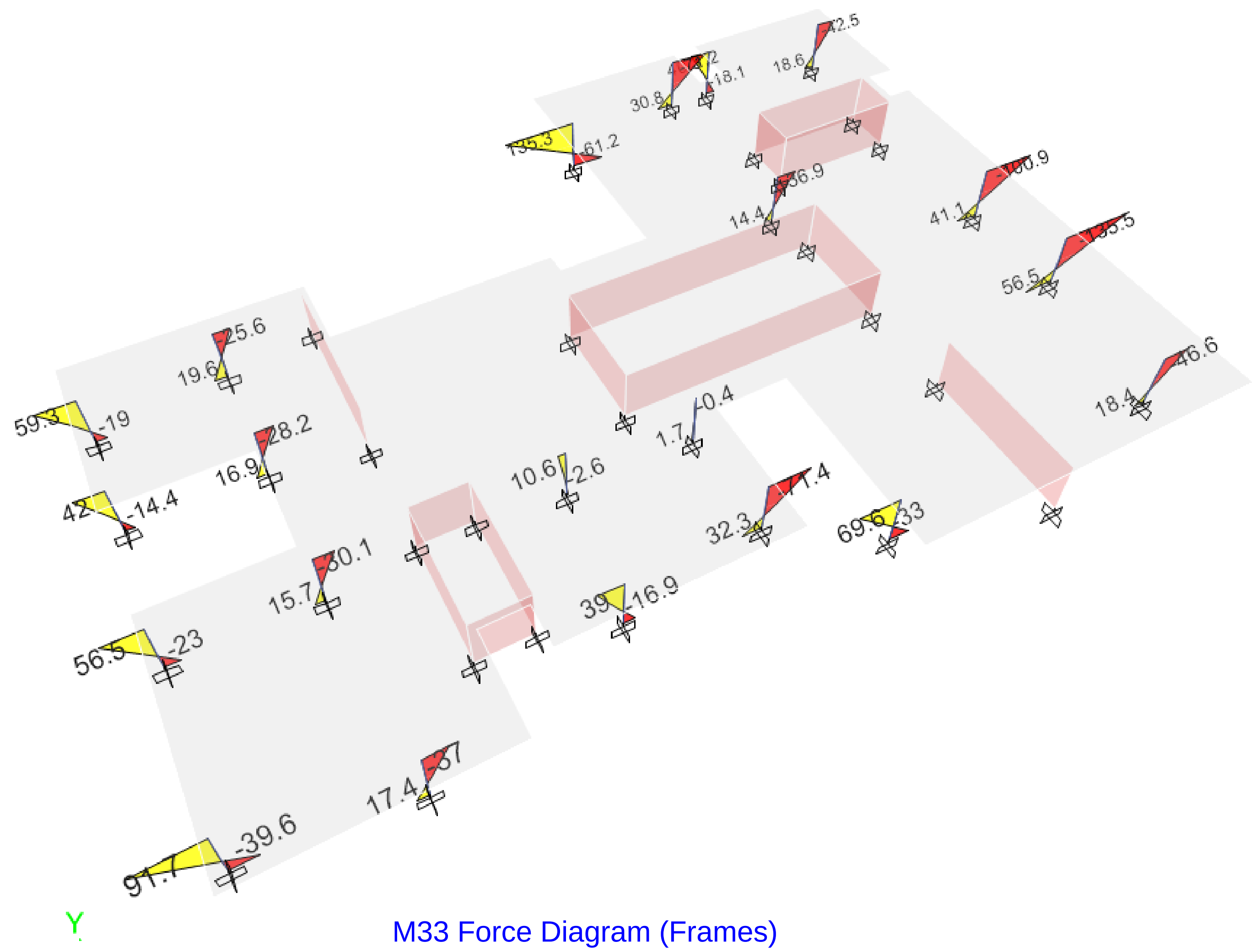
FEAKalc 3D



Axial Force Diagram

Under Self Weight

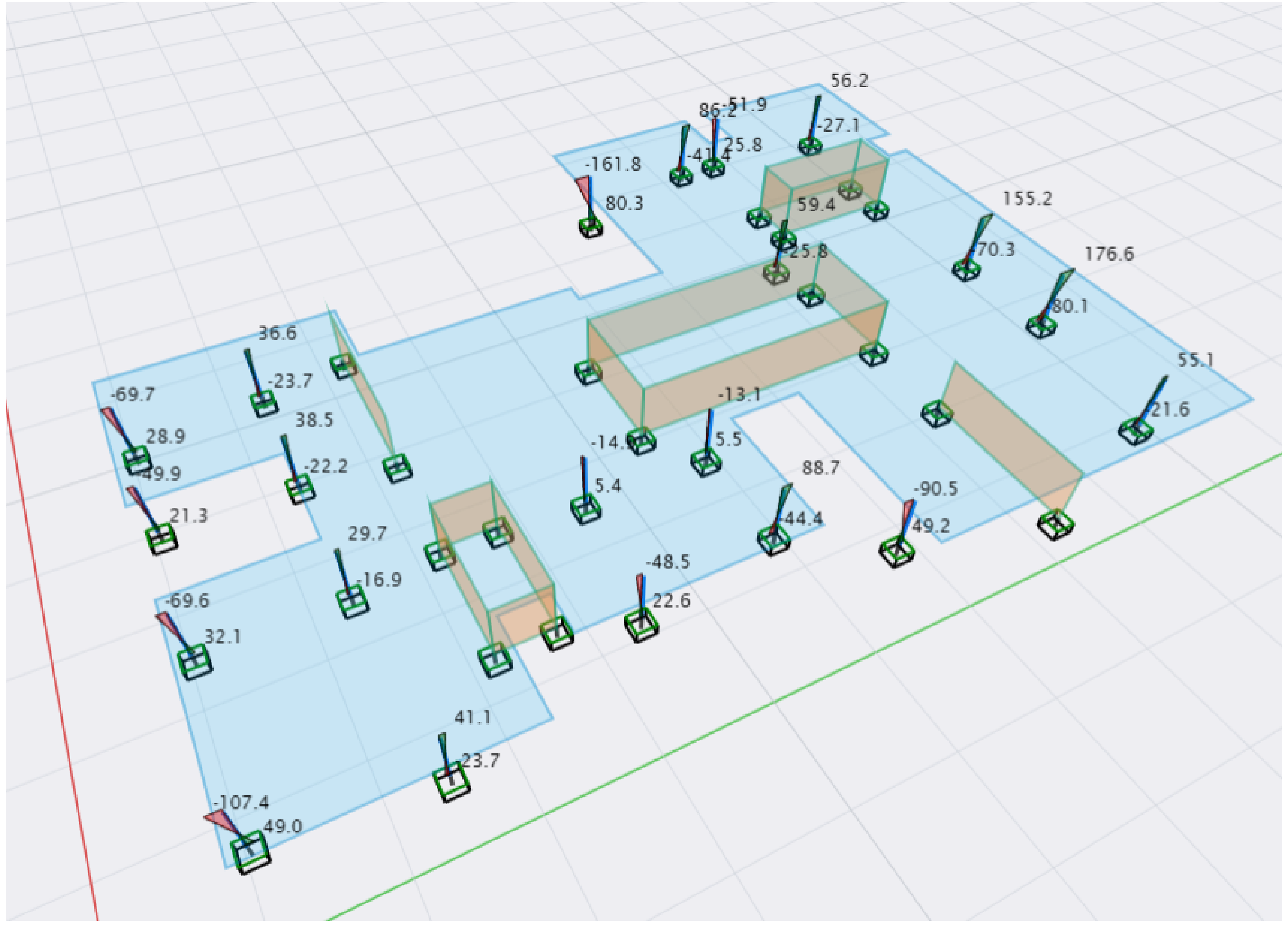
ETABS



M33 Force Diagram (Frames)

Under Self Weight

FEAKalc 3D



M33 Force Diagram (Frames)

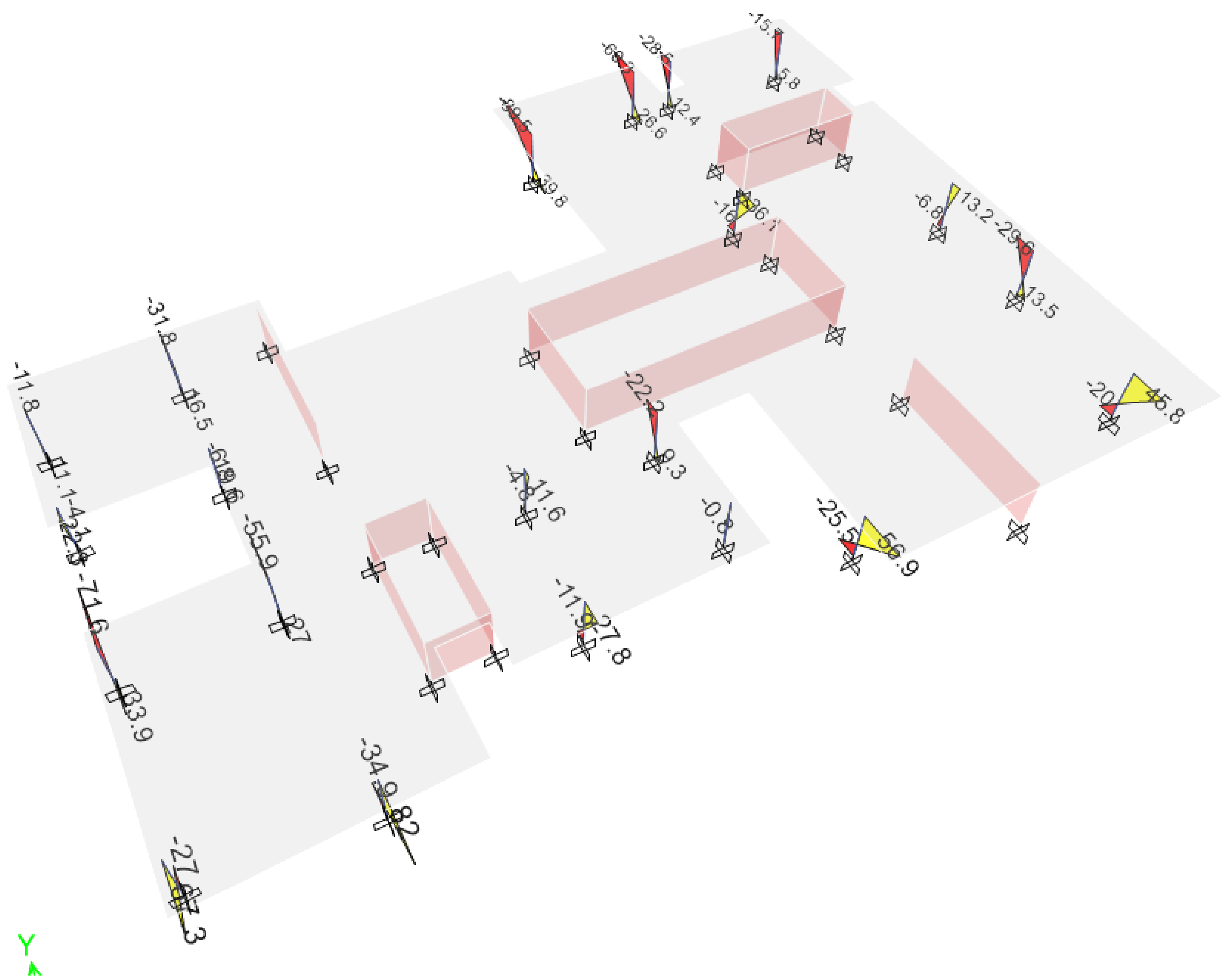
Under Self Weight

The comparison demonstrates good overall agreement between FEAKalc 3D and ETABS in the distribution and magnitude of frame end moments. Local differences remain at some slab–column connections, where FEAKalc generally predicts slightly different connection moments than ETABS. These differences are expected, as frame-to-shell interaction is influenced by the implementation of shell formulations, rotational compatibility, rigid connection modelling, mesh topology, and numerical result recovery. Despite these local variations, the overall structural behaviour, load paths, support reactions, and moment distributions show close agreement, confirming that FEAKalc provides engineering results consistent with those obtained from established commercial finite element software.

LEARNING CENTRE	Analysis Verification	Concrete Frame	By	RK
		FEAKalc 3D Verifications	Sheet No. 14	Date 7/26

ETABS

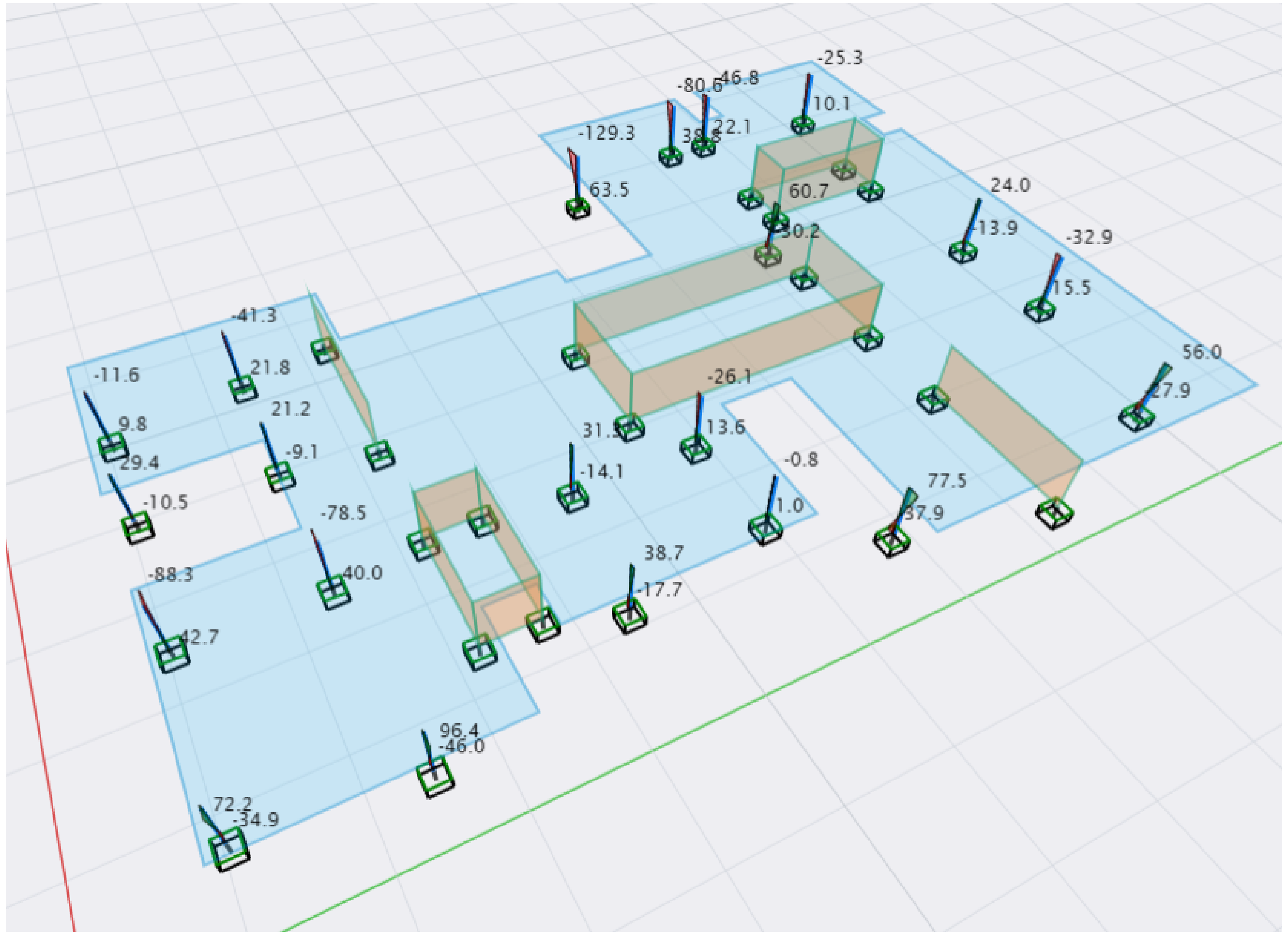
See note in page 11



M22 Force Diagram (Frames)

Under Self Weight

FEAKalc 3D



M22 Force Diagram (Frames)

Under Self Weight