

Bug Report No:4

AUTODESK BUILDING DESIGN SUITE ULTIMATE 2014
REVIT STRUCTURE & ROBOT STRUCTURAL ANALYSIS BOUNDARY CONDITION ISSUES

Definition of the problem:

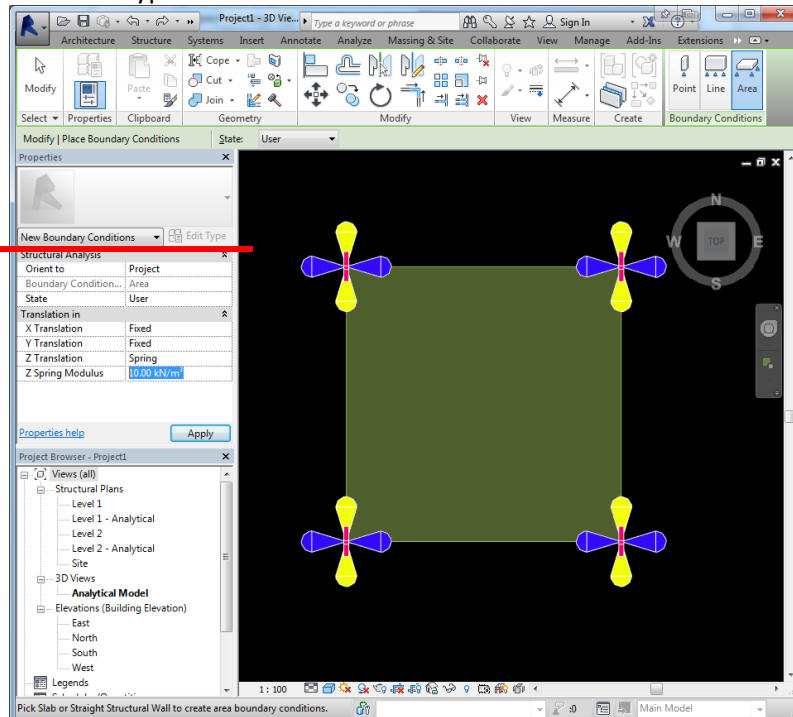
- 1- In Revit user cannot create distinct types of boundary conditions with unique names unlike other type based element such as columns, beams etc.
- 2- When transferred to Robot, for all foundation slabs “Elastic Foundation Coefficient” is taken as 1.02 t/m³ which produce results as an unintended extra spring support.
- 3- Z spring modulus given as boundary condition doesn't get accounted in “Soil reactions –K” calculation in Robot.
- 4- Boundary condition definition is not flexible enough, to let user define different values of spring modulus, inside the boundaries of a given slab. Such that Geotechnical engineers may provide regional values of subgrade reactions under the foundation slab. This requires division of foundation slab into many arbitrary parts.

Resolution suggestions:

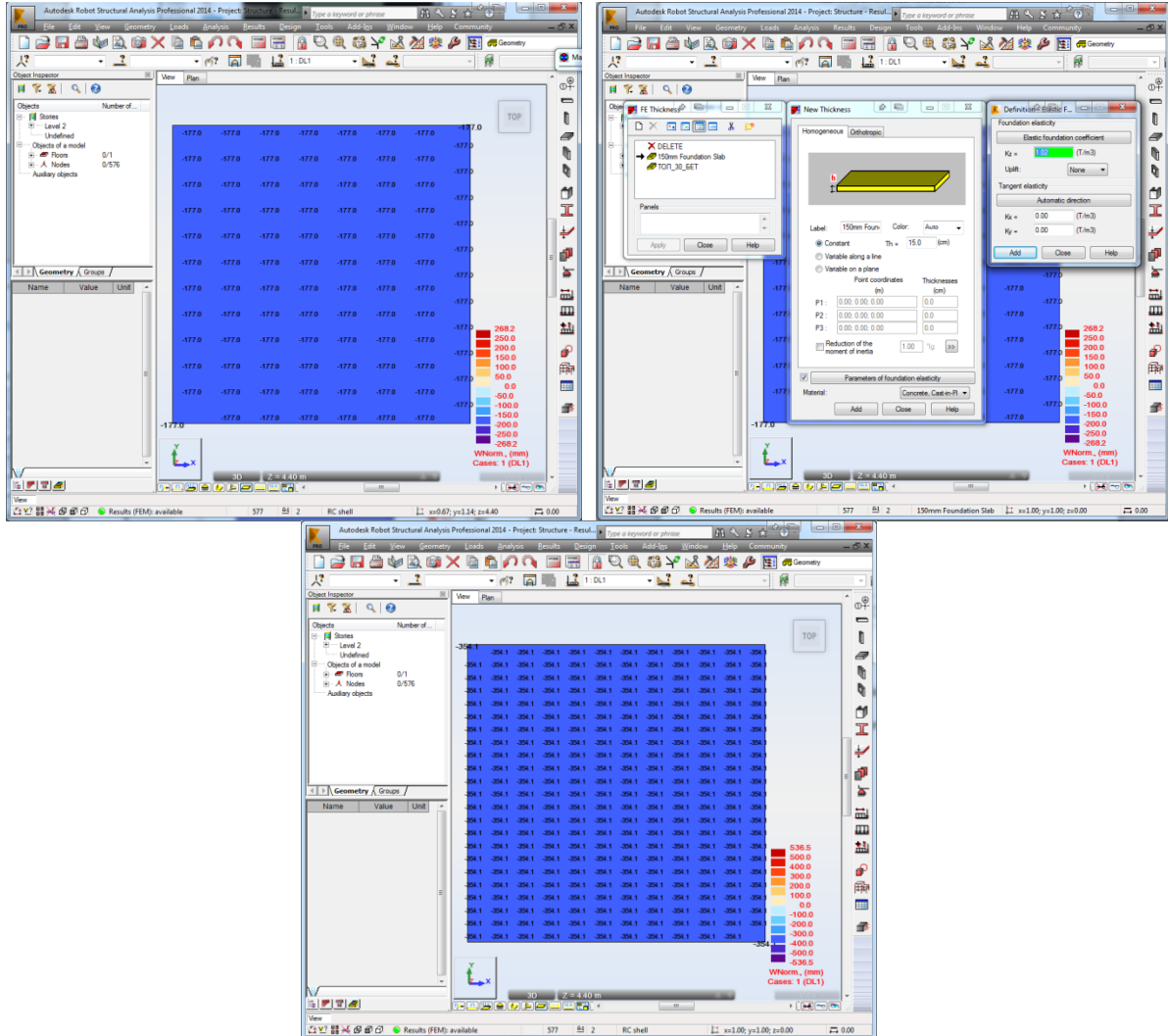
- 1- Boundary condition element should be enhanced in Revit and user could create more than one type with different names and support freedom conditions.
- 2- Default assignment of “Elastic Foundation Coefficient” should be “zero” t/m³
- 3- Best is: Revit Z Spring module is taken into account in Robot “Soil reactions – K” calculation. Or there should be a type property in Revit as “Elastic Foundation Coefficient” which can be assigned to slab elements separately. Like the analytical type properties such as “Absorptance” or “Roughness” of slabs.
- 4- Definition of Boundary conditions can be similar to non-hosted loads in which user can assign different values for arbitrarily defined regions of foundation slab.

Steps to recreate the problem:

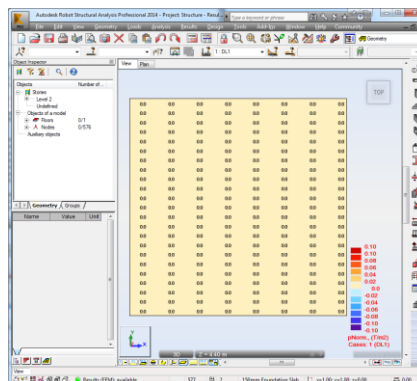
- 1- Select “Boundary Conditions” from Analysis tab click area or another one; there it's not possible to create a different type with a “*new name*” and condition.



- 2- In Revit create a slab of 1mx1m dimensions with otb. 150mm Foundation Slab
 - a. Assign an area boundary condition with fixed X Y translations and 10 kN/m³ Z Spring Modulus to the slab analytical model
 - b. Send model to robot and analyze: Initial analysis provides -177mm displacement which is wrong
 - c. Set the value of “Elastic Foundation Coefficient” to zero (“Thickness”-> “Foundation elasticity”) and re-run the analysis: now the result is true as 354.1 mm



- 3- Continue from the above state of the model and display “Soil reactions – K”, See that reactions are zero



4- Tedious process of creating (or worse: modifying) a foundation slab to assign regionally different spring moduli, may lead to many critical steps causing modelling errors. Division of floor/slab elements in Revit is not an easy task and at least cause information loss (such as loss of drafting elements loss of constraints and joining/attachment info of elements). Furthermore brings an extra analytical adjustment burden. (Revit developer team must provide a tool to divide slabs/floors not by recreating the contour sketch similar to the split tool for walls)

For example; check the below sketch that represents a real life Elastic Foundation Coefficient distribution of the ground soil. In its nature this is an iterative process where structural and geotechnical engineers pass each other results till the solutions converge to satisfy needs. However in order to model such a foundation Revit & Robot should provide more flexible modelling tools such as hosted or non-hosted area boundary conditions which can be sketched in free forms. Those boundary conditions should be taken into account in soil reaction calculations in Robot as stated on the above issue.

Below in Robot screenshot you can see how many different slab sections are needed to be created in order to assign relevant foundation coefficients. Assignments in Robot are panel based not finite element based as other analysis software packages. This makes the process very cumbersome; divide/re-sketch the slab in Revit and assign foundation coefficients in robot...

