



## WEBINAR

# Hybrid Mesh – An Innovative Approach to Perform Fast and Accurate Analysis in RS3

**Q1: What is the edge effect for the edge Hybrid elements? How many elements should we consider outside the area of interest for a stable result?**

A1: As we are applying constraints between 4 and 10 noded elements (explained in the webinar) the results would be continuous at the boundaries of these element types. So there is no edge effect at the boundaries of the hybrid zone. Regarding to the number of elements outside of the hybrid mesh, it really depends on the size and properties of the model.

**Q2: What is the number of DoF?**

A2: Each node in FEM, has a Degree of Freedom (DOF) meaning that, the actual unknown we need to describe a behaviour at that node. For example, if you are doing solid analysis, each node has 3 degrees of freedom for displacement, meaning that the node can move in all 3 directions. If you are doing only seepage analysis, we have only 1 degree of freedom at each node for pore pressure.

**Q3: What is the application of SRF-Max. displacement graph?**

A3: It shows you exactly when the failure happens at the model and at which SRF the yielding starts. Because sometimes your model may not converge due to some local failure or some numerical issues (or sometimes the number of iterations was not enough to get the convergence) but the failure of the slope hasn't been initiated yet. In these cases, if you check the SRF-MaxDisp graph, you can easily realize if the calculated critical SRF is valid or not.

**Q4: What will be a maximum number of iterations with hybrid mesh?**

A4: The maximum number of iteration is independent of the having hybrid mesh or not. But we recommend to keep the default values we have in project setting and go with 500.

**Q5: Can we use this method in finite difference as well?**

A5: Yes you can use same approach in finite difference as well. But RS3 is a FEM package and we don't deal with finite difference.

**Q6: How much time would we save if we used the hybrid mesh versus entirely 10-noded?**

A6: This was shown during the webinar. For the slope stability analysis, the speed of calculation was 5 times faster using hybrid mesh in comparison with 10-noded elements.

**Q7: Can we have a graded 10-noded mesh surrounded by a uniform 4-noded mesh? Or do they both have to be graded?**

A7: Very good question. Yes it can be done, if you assign a uniform mesh for the whole model, and then define a refinement region at the location of the hybrid mesh, and set the mesh setting for the refinement region to "Extent" (not to be uniform). But we don't have Gradation control and it's all automatic. But I take your comment as a great suggestion to be implemented in RS3 in future. Thanks!

**Q8: As we have the hybrid which seems to give the same accuracy as the 10 node calculation, for which situations would you suggest to use the 10 node simulation?**

A8: We set the hybrid mesh at the locations where we expect to see the failure or where the displacement gradient is high. If we cannot find these locations in advance, you can use 10 noded elements everywhere, but we don't recommend it since it might be slow. So the better solution is to use a coarse 4 noded mesh, detect the areas where we can have failure, and then assign hybrid mesh on those locations. For complex models, these two steps would be way faster than just running a model with 10 noded elements.