



This Group ▼

Search this group...



My Dashboard

Ansys Campus ▼

Ansys Solutions ▼



Python commands to Activate/Deactivate a displacement BC for a specific load step

Vincent · 2 months ago · Question

[Structures | Customization](#) / [Forums](#) / [Ansys Mechanical Scripting](#)

Hi,

I would like to apply a displacement boundary conditions, which will be defined with tabular data (0 in X,Y,Z direction) for the initial load step, but deactivated for the following load step (which can be done by RMB click on the load step number -> 'Activate/Deactivate at this step'). What would be the Python equivalent for that?

Regards,
VB

✓ Best Answer

```
Landon disp0 =  
ExtAPI.DataModel.GetObjectsByType(Ansys.ACT.Automation.Mechanical.BoundaryConditions.Displacement)[0]  
disp0.SetActivateAtLoadStep(1,True)  
disp0.SetActivateAtLoadStep(2,False)
```

about 2 months ago · 1 Like

👍 Like

📡 Subscribe

▼ More

Comments



Comment on post



Post

Landon

about 2 months ago

✓ Best Answer



```
disp0 =  
ExtAPI.DataModel.GetObjectsByType(Ansys.ACT.Automation.Mechanical.BoundaryConditions.Displacem  
ent)[0]  
disp0.SetActivateAtLoadStep(1,True)  
disp0.SetActivateAtLoadStep(2,False)
```

1 Like