

# ANSYS - Tips and Tricks

- [Accessing ANSYS](#): This tip shows students the three main ways of accessing ANSYS
- [Navigating ANSYS Mechanical 2019 R2 Interface](#): This tip shows some of the changes in the new interface and tips on how to not be confused by the changes.
- [Time Dependent Boundary Condition](#) : This tip shows how to specify a time-dependent temperature boundary condition by reading in values from Excel.
- [Activate Academic Research License](#) : This tip shows how to switch from a teaching license to a research license if you have purchased both licenses. The teaching license has a limitation on the maximum number of nodes in your model whereas the research license doesn't put a limit on the maximum number of nodes.
- **Set Number of Cores**: To set the number of cores in ANSYS Mechanical, go to Tools > Solve Process Settings > Advanced and change *Max number of utilized processors*. You will need a valid HPC license to use more than 4 cores. To maximize the amount of memory available for a solve, close memory hogs and start-up programs before hitting *Solve*.
- [Fatigue Analysis](#) : This tip shows how to perform fatigue analysis in ANSYS Mechanical.
- [Hydrostatic Pressure](#) : This tip demonstrates how to use the Hydrostatic Pressure load to determine the water level.
- [Auto Constraints](#) : This tip demonstrates how to turn on Auto Constraints in DesignModeler, which is not turned on by default in ANSYS 15.0.
- [Moving or translating a geometry in DesignModeler \(ANSYS' Old Geometry Engine\)](#)
- [ANSYS Installation Guide for Mac OS X](#)
- [ANSYS Installation for Windows](#)
- [Running FLUENT on a Remote Server](#)
- [How to Compile UDFs](#)
- [How to interpret principal stresses for 2d plane stress cases in ANSYS Mechanical? Why is one principal stress not zero everywhere?](#)
- [How to Vary Material Properties along an Object](#)
- [Specifying spatially varying loads in ANSYS Mechanical: See this blog post from another site](#)
- [Introduction to Command Object Post-processing](#)
- [Importing Surfaces from Solidworks into ANSYS](#): this tip will show you how to import just a surface into DesignModeler using Solidworks
- [Using Force Reaction Probes on Contact Surfaces](#)
- [Accessing Selection Information window in ANSYS](#)
- [Creating a Surface in DesignModeler from Imported Geometry and Combining Edges](#)
- [Connecting to Swanson Lab using Remote Desktop in Windows](#)
- [Material Property Specification Fix](#)