• **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**