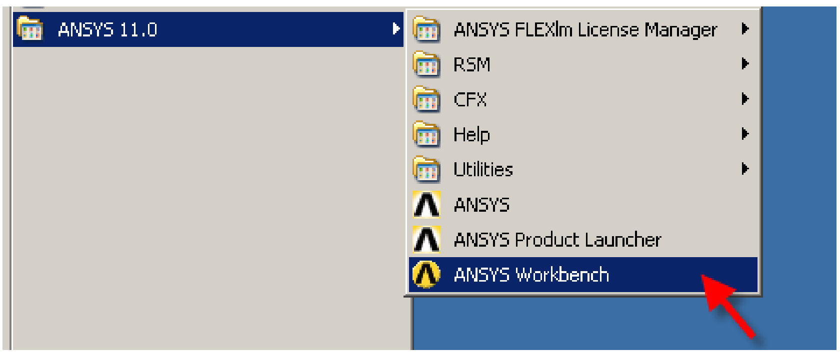
Instructions on How to Access ANSYS CFX/CFX-Mesh Tutorials

There are two parts of tutorials offered by ANSYS to teach you how to get CFX mesh in **ANSYS Workbench** and how to conduct a CFD analysis in **ANSYS CFX**. The first part is in the **ANSYS Workbench** help files, which teaches you how to build up geometry and get CFX meshes. The second part is in the **ANSYS CFX** help files, which teaches you how to conduct CFD analysis based on the mesh you built in the first part of tutorials. You can also skip the geometry and mesh building part by using the mesh files located at:

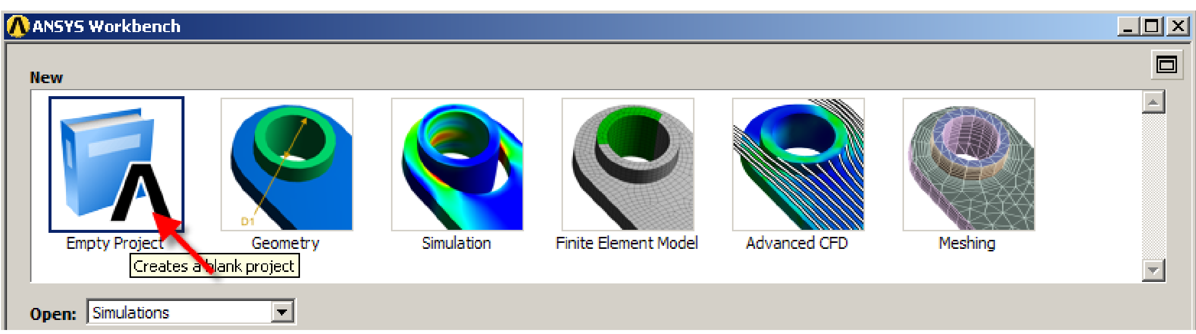
C:\Program Files\ANSYS Inc\v110\CFX\examples

ANSYS WORKBENCH CFX-Mesh Tutorials

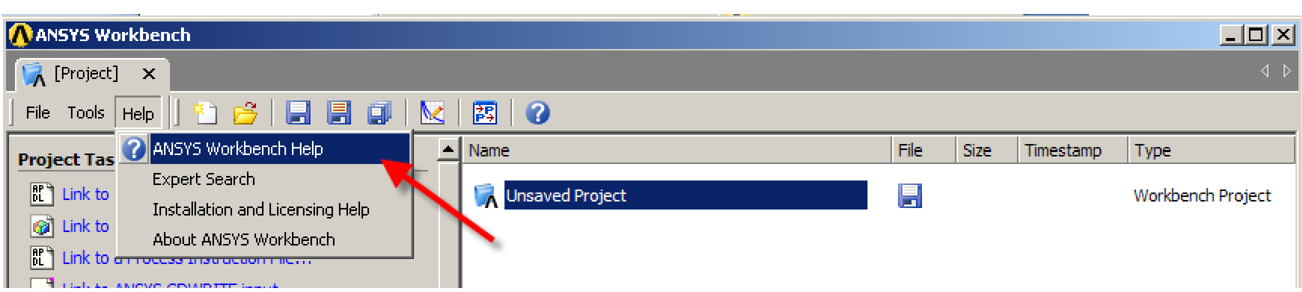
1. Open "**ANSYS Workbench**" by clicking "**START**" -> "**ANSYS 11.0**->"**ANSYS Workbench**"



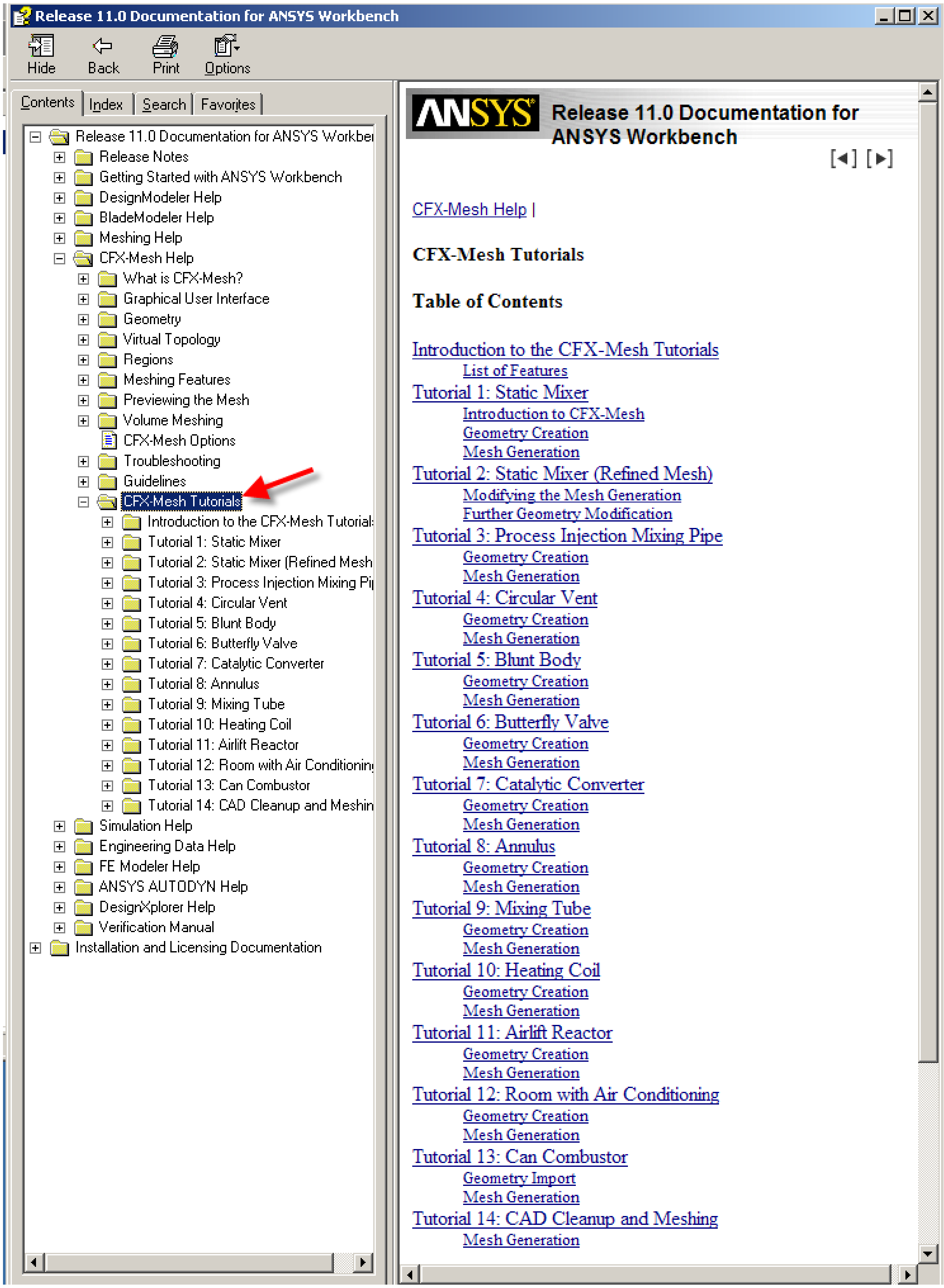
2. Open a new "**Empty Project**" in ANSYS Workbench.



3. Click on "**Help**"->"**ANSY Workbench Help**"

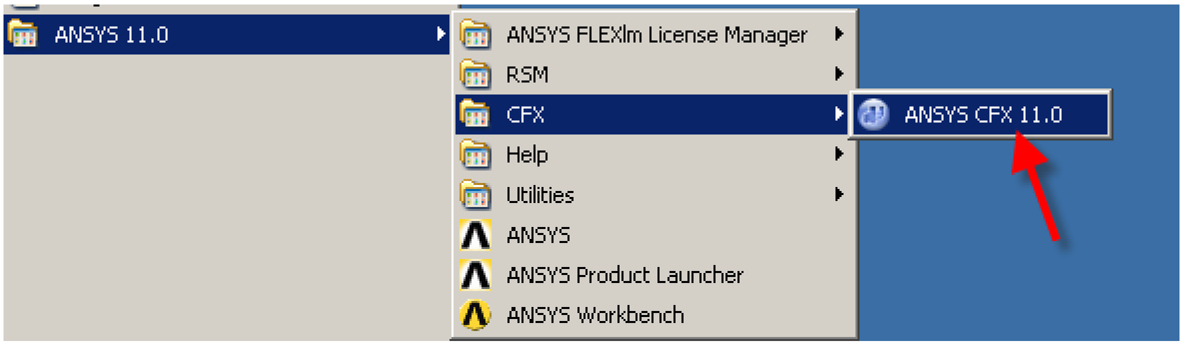


4. Look for "**Release 11.0 Documentation for ANSYS Workbench**"->"**CFX-Mesh Help**"->"**CFX-Mesh Tutorials**" in the "**Contents**" tab on the left.

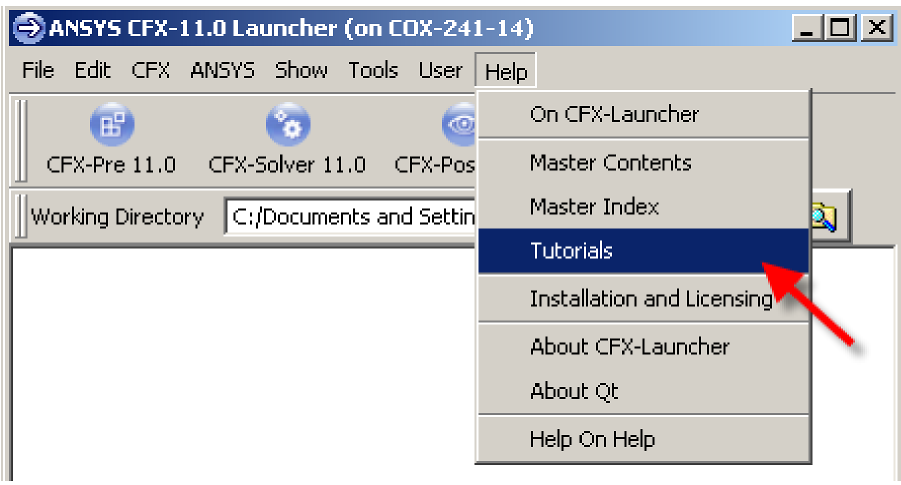


ANSYS CFX Tutorials

1. Open “**ANSYS CFX-11.0 Launcher**” by clicking “**START**” -> “**ANSYS 11.0**”->”**CFX**”->”**ANSYS CFX 11.0**”



2. Since the ANSYS CFX-11.0 Launcher is opened, click on “**help**”->”**Tutorials**”



3. Look for “**ANSYS CFX Tutorials**” in “**Contents**” Tab on the left.