

## ANSYS Workbench – a short guideline

*This document is a guideline on how to use ANSYS Workbench 2.0 as a tool to solve the second computer workshop in the course SE1025 FEM for engineering applications. You are supposed to have read the problem descriptions before proceeding with this document.*

ANSYS Workbench is a common platform for solving engineering problems. Typical tasks you can perform in Workbench are:

- Importing models from a variety of CAD systems.
- Conditioning models for design simulations using the DesignModeler.
- Performing FEA simulations using Simulation.
- Optimizing designs using DesignXplorer or DesignXplorer VT.

The underlined words above are the names of different processors within ANSYS Workbench. Basically, you will use the DesignModeler to create the geometry and the Simulation to set up the materials, FE-mesh, loads and boundary conditions, solve the problem and analyse the results. The standard interface ANSYS Classic (used in the first computer workshop) is still the core of ANSYS. ANSYS Workbench is a new modern interface with more up to date functions such as, for example, the integration of CAD geometries.

### **DesignModeler**

DesignModeler is designed to be used as a geometry editor of existing CAD models. DesignModeler is a parametric feature-based solid modeler designed so that you can intuitively and quickly begin drawing 2D sketches, modeling 3D parts, or uploading 3D CAD models for engineering analysis preprocessing.

### **Simulation**

Use the Workbench Simulation module to define your model's environmental loading conditions, solve the simulation, and review results in various formats depending on the type of simulation.

# Problem 1

## NOTE:

It is of vital importance that you choose a working directory with enough space available!  
Otherwise the whole analysis will fail!

In this section, the modeling and analysis of the thermo elastic problem will be described. Start the **ANSYS Product Launcher**. Select a working directory for storing your model data and launch **ANSYS Workbench**. You will see the software outfit as Fig. 1.

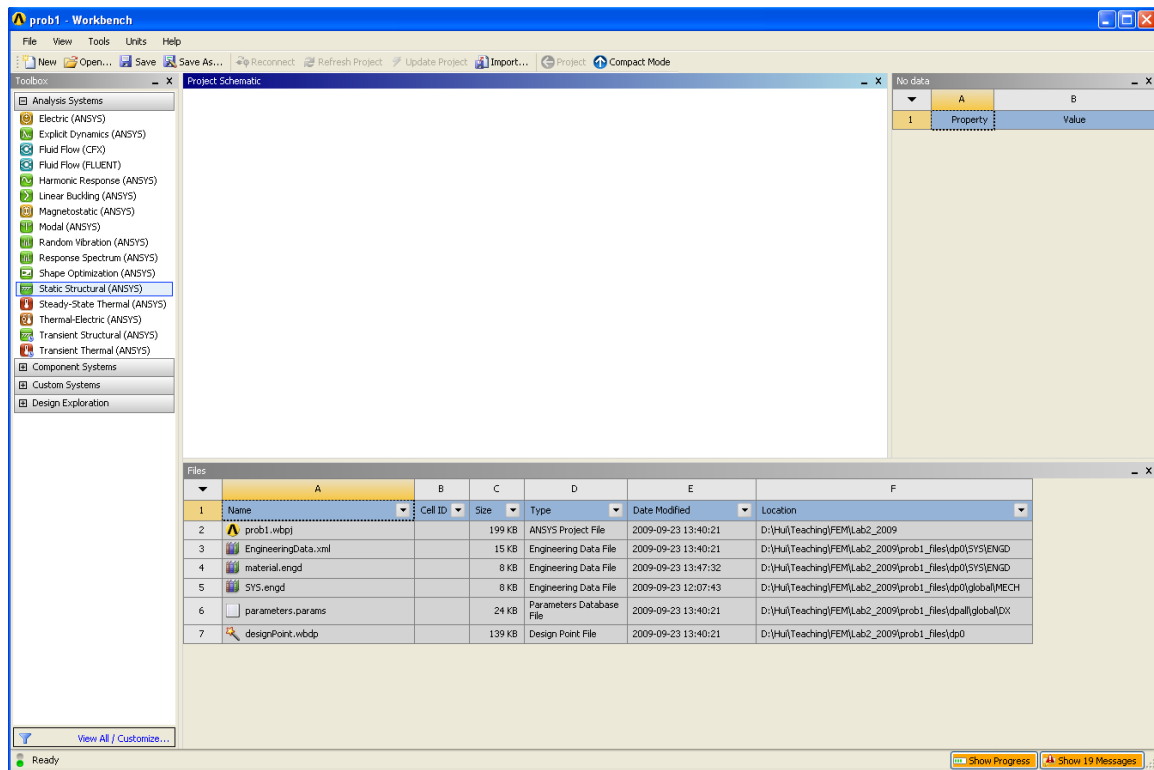


Figure 1 Software outfit

1. Drag **Static Structural (ANSYS)** tab from **Analysis Systems** of **Toolbox** window to the **Project Schematic** window. Now, your static structural analysis model should be in the **Project Schematic**, as Fig.2 shows.

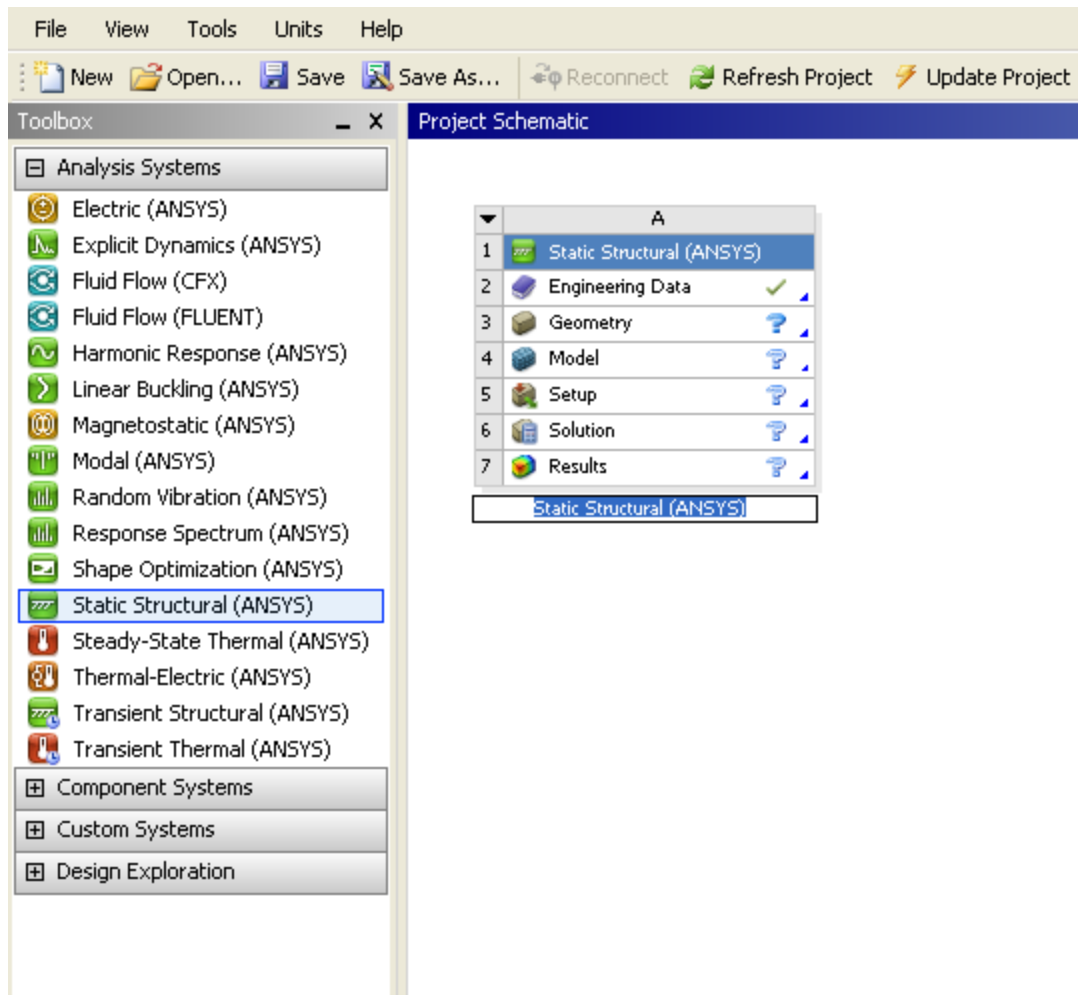





Figure 2 Workbench Project Mode

2. Double click  Engineering Data , then you will go to Engineering Data atmosphere (as Fig.3). At **Properties of Outline Row 3 window**, you can fill in the required material data, and click the data at column E with tab  , so that the material data you chose will show up in the **Parameter Set** afterwards. But in **Static Structural** mode, you only can define some parameters. The thermal conductivity you have to define it under **Steady-State Thermal** mode later. After filling parameters, click  Return to Project to go back the Workbench project mode.

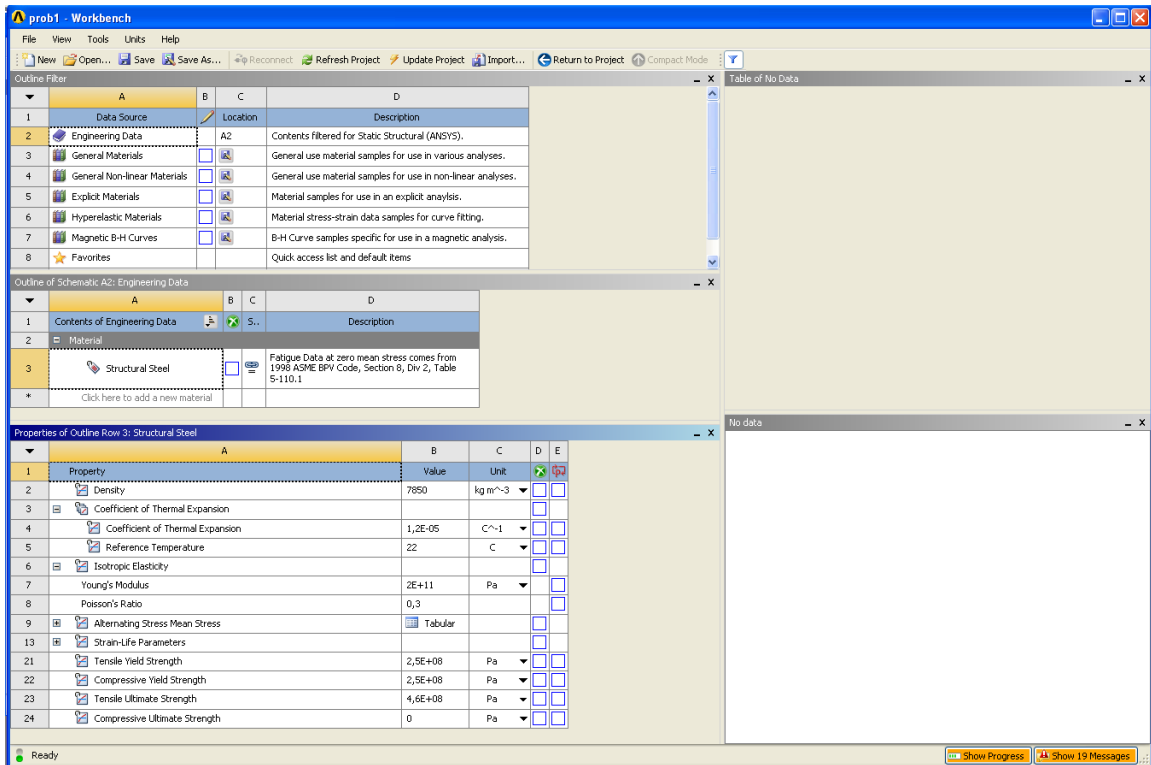


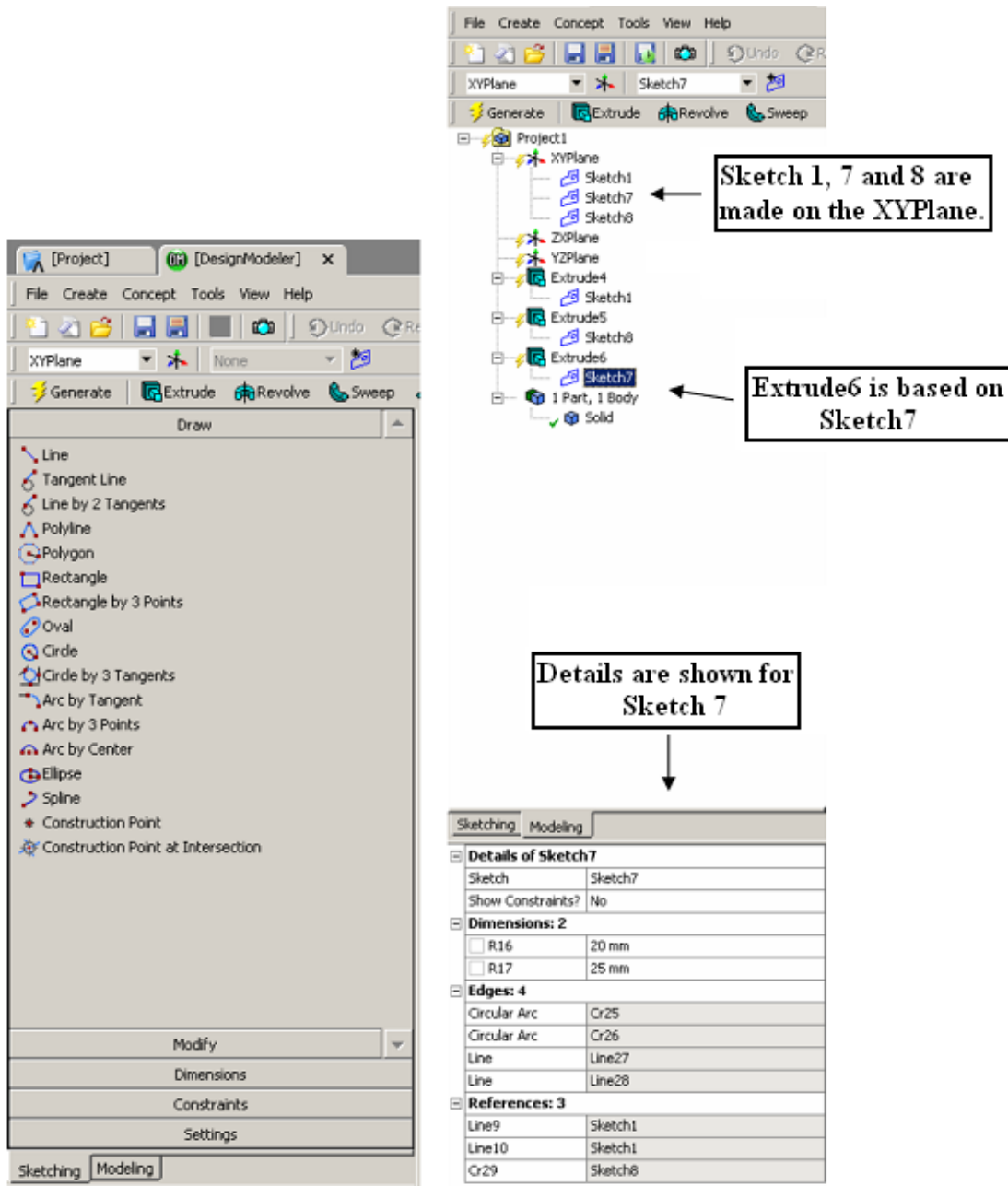


Figure 3 Engineering Data Mode

3. When you finish filling the material data, then you can start to build geometry. Double click  **Geometry**, and you will activate **DesignModeler**. A 3D-geometry in DesignModeler is defined on basis of 2D-sketches. The sketches are used to extrude, revolve or sweep a 3D shape. In Fig.4 some basics are described. We will first sketch the base of the pipe and fin. The faces will then be used to create the volume by extrusion.

- Click the **Modeling** tab and place the cursor on **XYPlane** in the model tree. Create a new sketch by clicking .
- Click the **Sketching** tab. Under the **Drawing** tab, select **Rectangle** and draw a rectangle in the drawing window.
- On the **Dimensions** tab, select **General** and dimension the two sides of the rectangle: the height and the width. Clicking a side of the rectangle will allow you to define the length of this side. In the "Details View" window, change the dimensions to 100 mm.
- Create another sketch on the **XYPlane**. Select **Arc by Center** and draw a quarter of a circle located inside the triangle, the center point of the arc should be placed on the bottom left corner of the rectangle. Repeat this once so that you obtain two arcs with different diameters. Use then dimension to set the radii to 25 and 20 mm respectively. Connect the end points of the arcs with two short lines of 5 mm each.

- Create the third sketch on the same plane. This sketch should consist of one single circle coincident with the two arcs above. The radius should be 25mm.



**Figure 4: Sketching and Modeling modes. On the left, the sketching tab is shown. Here you can use different 2D drawing functions such as line, rectangle and circle for drawing a 2D sketch. On the right, the modeling tab is shown. Here you can find the model tree. In this example there are 3 sketches, 3 extrusions and one solid body. Each extrusion is based upon a sketch and each sketch lays on a plane, in this case the XY plane.**

When you are done with the sketches, the result should be something like Fig.5.

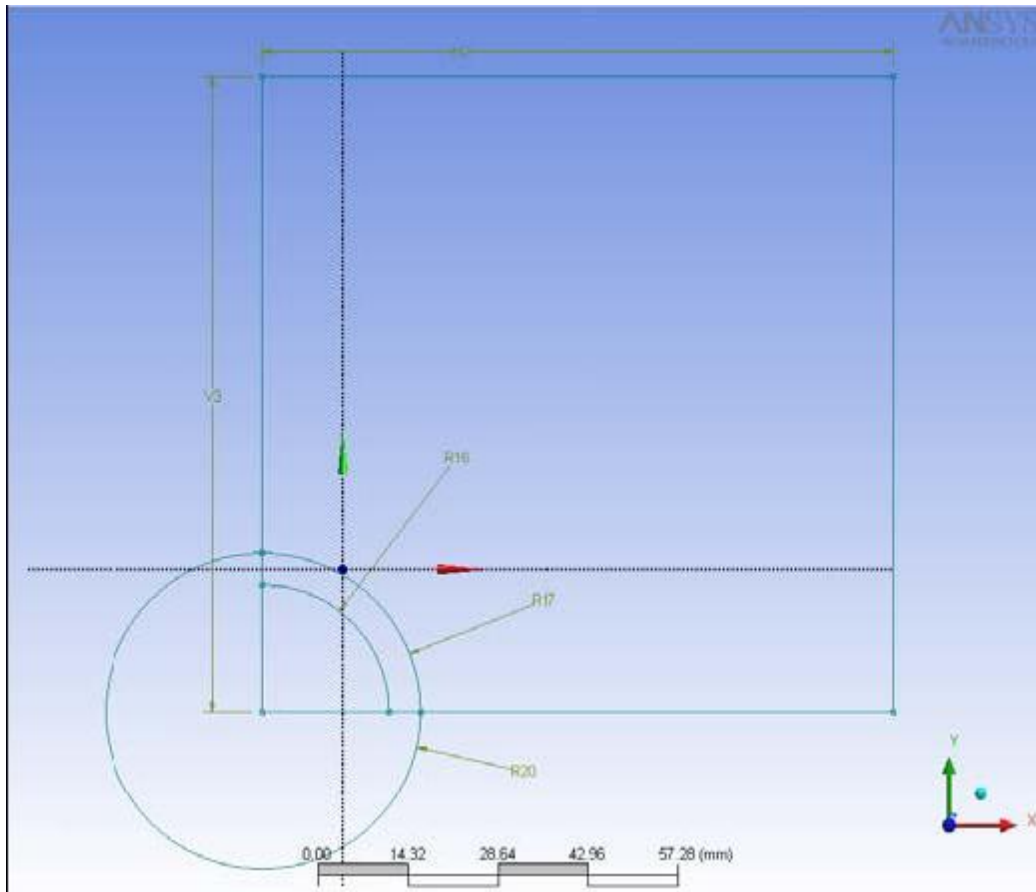
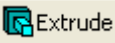

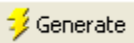



Figure 5: The three sketches with dimensions

- You will now use the sketches to create extrusions. First you will create the base plate for the fin. Select the first sketch and click the Extrude button  Extrude
- In the details window below the modeling tree, check that the **Operation** is **Add Material** and that the **Depth** is 5 mm. To create the extrusion, click  Generate
- To create the hole for the pipe, create an extrusion based on the third sketch. In the details window, check that the **Operation** is **Cut Material** and that the **Type** is **Through All**. To create the extrusion, click  Generate
- The pipe is based on the second sketch. In the details window, check that the **Operation** is **Add Material**, the **Depth** is **50 mm** and that **Merge Topology** is turned on. To create the extrusion, click  Generate

The geometric model is shown in Figure 6.

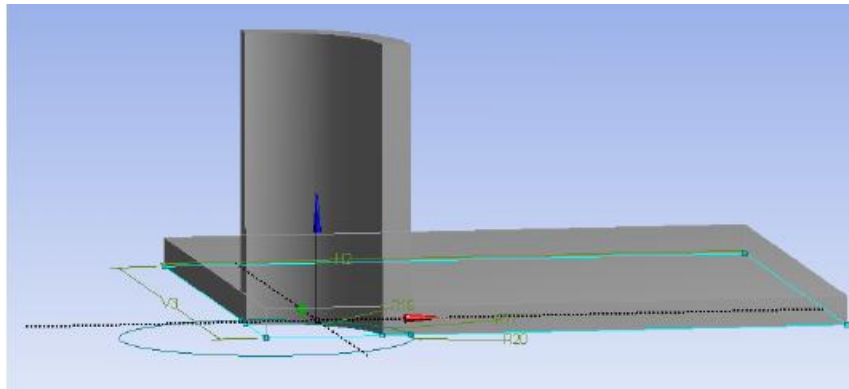



Figure 6: Geometry model.

4. Leave **DesignModeler**, go back to Workbench project mode. The geometry will now automatically be imported into the simulation processor within Workbench. But don't forget **save everything** and then do further simulations.
5. And then the following analyze will be done in two steps. The first step is to calculate that result in mechanics analyses, the second step is to add the temperature distribution of this cooling flange with results from mechanical analyze. We start with the mechanical problem. To start analyze, you need to double click  Model at Workbench Project mode. Then you will go to simulation mode as Fig. 7.

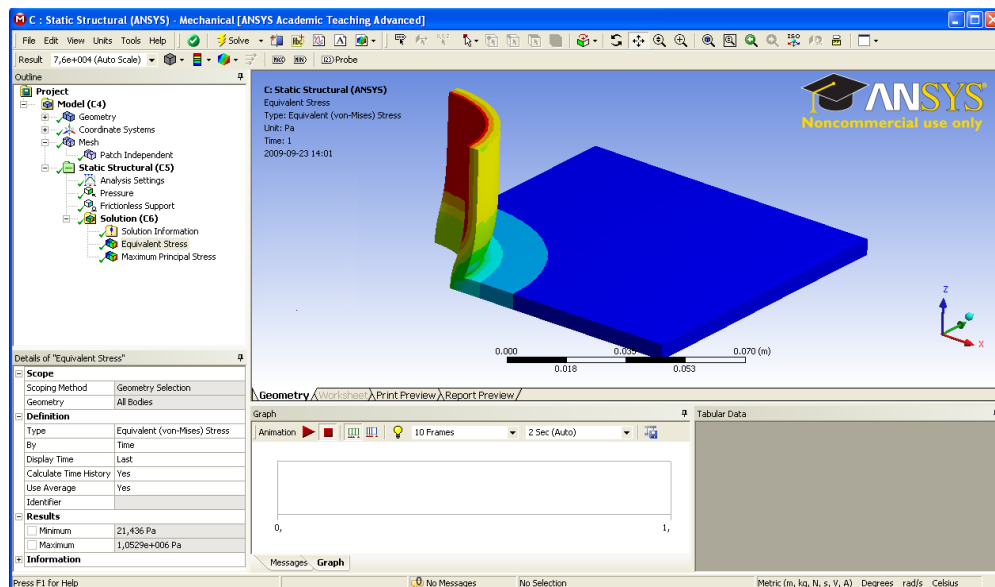







Figure 7

6. Following instruction describes how to execute the mechanical analysis:

- Mesh your model by right click  Mesh , and choose **Generate Mesh**
- Add boundary conditions. Select all symmetry surfaces. Right click on **Static Structural**. Under insert select **Frictionless Support**. To apply the loads, select the inside of the pipe. Right click on **Static Structural**. Under insert select **Pressure**. Enter the value of the internal pressure.
- To solve the problem click the solve button. To see Von-Mises effect stress, right click **Solution**. Choose **Insert > Stress > Equivalent (Von Mises)**.write down the result.  
**Save** your simulation.

7. Thermal-Mechanical Analysis:

- Go back to Workbench project window, and right click  Setup and choose **Transfer Data from New > Steady-State Thermal (ANSYS)**. Then a Steady-State Thermal project has been built which also related to the same geometry and model of Static Structural project. See Fig. 8
- Double click  Engineering Data under **Steady-State Thermal** project, add thermal conductivity parameter and choose it into parameter set by click . Click return to project button.
- Double click  Model under **Steady-State Thermal** project
- Now, you can add thermal condition to the simulation. Highlight **Steady State Thermal** in **Project Tree**. Define convection by selecting **Convection** under insert.
- Solve this problem and read results of temperature. Add **Temperature** into **Solution**.
- Solve the Static Structural problem again with von-Mises effect stress results. Write down the result. **Save** your simulation.

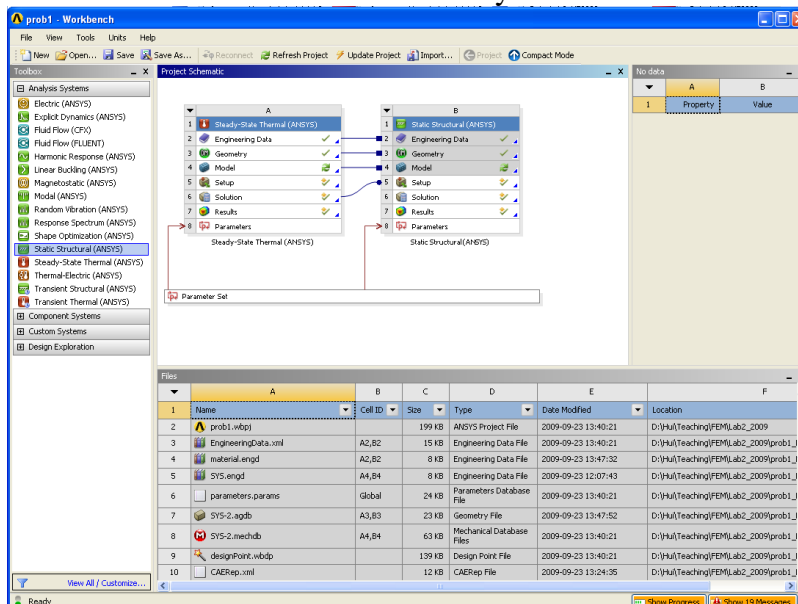


Figure 8



## Problem 2

This problem you are supposed to do, more or less, on your own. The pressure vessel from the problem description is axisymmetric, so you will only need to model  $\frac{1}{4}$  of the 2D cross section of the 3D pressure vessel:

1. First of all, mark **Geometry** under **Workbench** project mode. And at **Properties of Schematic A3: Geometry** window, expand **Advanced Geometry Options**. Change **Analysis Type** to 2D
2. Define the **material**.
3. Draw a sketch of the 2D model ( $\frac{1}{4}$ ) in DesignModeler. For 2D FE-models in Simulation, you will need to create a surface from your sketch. When you're done with your sketch, on the menu click on **Concept – Surfaces From Sketches**.

### NOTE:

*The option **axisymmetry** assumes that a 3-D model and its loading can be generated by revolving a 2-D section  $360^\circ$  about the y-axis. The axis of symmetry must coincide with the global y-axis. The geometry has to lie on the positive x-axis of the x-y plane. The y direction is axial, the x direction is radial, and the z direction is in the circumferential (hoop) direction.*

4. The option for axisymmetry can also be found in the model tree on the left. In the “Details of...”-window for your **Geometry, Definition**, options for **2D Behavior** are Plane Stress, Plane Strain or Axisymmetry. Choose **Axisymmetry**. Mesh the 2D surface model.
5. Define the **internal pressure**.
6. Define the **symmetry boundary conditions** and add at least one target result.
7. Solve and analyze the results.