**Fluid Network Analysis**

**Introduction**

This tutorial was created using ANSYS 16.1. The purpose of this tutorial is to determine pressure at nodes and flow rates in pipes for given fluid flow network.



**Define the Preferences**

* Select **Preferences**.
* The following window will appear. Select ‘ANSYS Fluid’ as shown.



**Define the Element Type**

* Select **Preprocessor>Element Type>Add/Edit/Delete**.
* The following window will appear. Select ‘Add’.
* Use **ANSYS Fluid>3D Viscous Link 138**.





**Define the Real Constants (Radius and length of each pipe)**

* Select **Preprocessor>Real Constants> Add/Edit/Delete**.
* The following window will appear. Select ‘Add’.
* Then select ‘Type 1 FLUID138’ and click ‘OK’.
* Define the size of the first pipe with DIM1 and DIM2 are 7.5e-3 and 7.5 separately.
* Repeat this process for another three times for other pipes 2, 3, and 4 as shown.









**Define the Material Properties**

* Select **Preprocessor>Material Props> Material Models**. The following window will appear.
* Select **Material Model Number 1>Fluids>Viscosity**.
* The VISC value is 8e-4.







**Define the Nodes**

* Select **Preprocessor>Modeling>Create>Nodes>In Active CS** and the following window will appear.
* Define the location of the first node with coordinate of (0, 0) and click ‘Apply’.
* Repeat this process for another three times for Node 2, 3, and 4 with coordinates of (0, -7.5) (10, 0) (10, -7.5) separately.



**Define the Elements**

* Select **Preprocessor>Modeling>Create>Elements>Auto Numbered>Thru Nodes**. Select Node 1 and Node 2 and click ‘OK’ so the Element 1 is defined.
* Select **Preprocessor>Modeling>Create>Elements>Elem Attributes** and the following window will appear. Choose Real constant set number ‘1’ for the Element 1.
* Repeat these processes for another three times. Connect Node 1 and Node 3 for Element 2, Node 1 and Node 2 for Element 3, and Node 3 and Node 4 for Element 4. The Real constant set number is 2, 3, and 4 separately.





**Define the Loads**

* Select **Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Pressure DOF>On Nodes**. Choose Node 4 and the following window will appear. Use PRES Pressure value ‘0’.



* Select **Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Flow>On Nodes**. Choose Node 1 and the following window will appear. Use FLOW Load value ‘0.16e-4’.



**Define the Analysis Type**

* Select **Solution>Analysis Type>New Analysis** and the following window will appear. Choose ‘Steady-State’.



**Solve the problem**

* Select **Solution>Solve>Current LS**.

**Check the results**

* Select **General Postproc>Plot Results>Contour Plot>Nodal Solu** and the following window will appear. Choose **Nodal Solution>DOF Solution>Pressure** and Click ‘OK’.
* Select **General Postproc>List Results> Nodal Solution** and the following window will appear. Choose **Nodal Solution>DOF Solution>Pressure** and Click ‘OK’.







