



## Pressure flow

© 2012 TechnoLogismiki

# WHITEPAPER

[www.technologismiki.com](http://www.technologismiki.com)

 **TECHNO logismiki**

Advanced Technical Software

5 Imitou str, 15561, Cholargos, Athens, Greece  
tel: ++30 210 65 64 147 - fax: ++30 210 65 48 461  
[www.technologismiki.com](http://www.technologismiki.com) - [info@technologismiki.com](mailto:info@technologismiki.com)

# Table of Contents

## Chapter I About the whitepaper

1 Purpose.....	4
2 Software.....	4

## Chapter II Steps

1 Step 01: New Project.....	6
2 Step 02: Add Solution.....	6
3 Step 03: Data input.....	7
4 Step 04: Calculations.....	9
5 Step 05: Printing.....	9

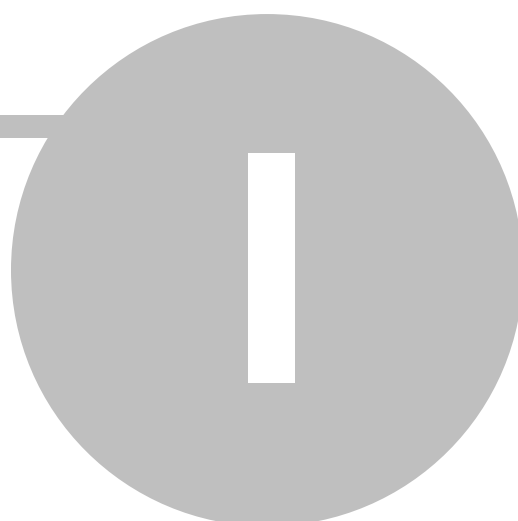
## Chapter III Help

1 Technical support.....	12
--------------------------	----

<b>Keyword Index</b>	<b>0</b>
----------------------	----------

# Chapter

---



# 1 About the whitepaper

## 1.1 Purpose

The purpose of this example is to calculate the flow rate of a circular section under pressure. The section has the following properties:

Property	Value
Pipe length	200 m
Diameter	0.350
Pipe slope	0
Pressure difference	20 atm
Fluid	Water 20°C
Friction coefficient	0.02

For turbulent flow ( $Re > 4000$ ) the Swamme – Jain formula will be used, for laminar flow ( $Re < 2000$ ) the Reynolds number will be set equal to  $64/Re$ , while for transitional flow the program will perform an interpolation using the Moody diagram.

## 1.2 Software

In order to complete the example successfully, the following software is required:

- Hydraulic Solver v11.0.

Later versions of the aforementioned software may be incompatible with the structure of the example as it is presented herein.

# Chapter

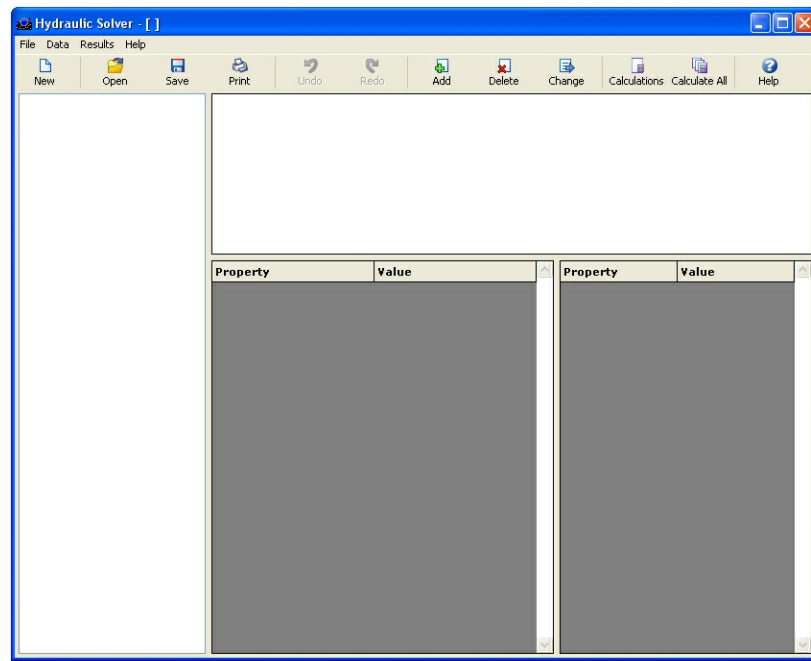
---



## 2 Steps

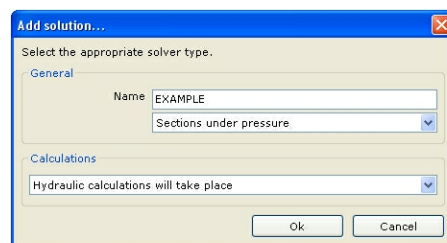
### 2.1 Step 01: New Project

Select **New Project** from the **File** menu. The program will remove any data from memory and prepare to start a new project:



### 2.2 Step 02: Add Solution

Select **Add Solution** from the **Data** menu. The following form appears:



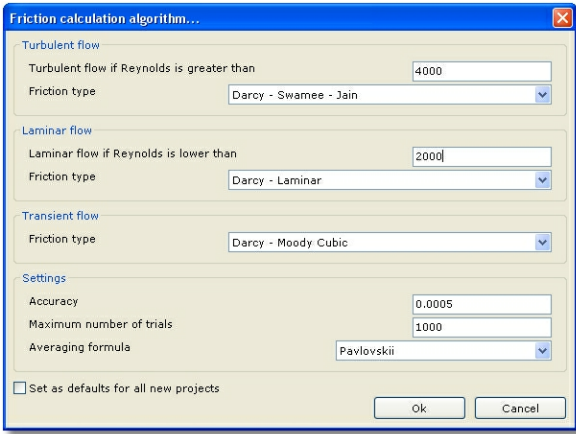
Enter the following data:

- **Name:** Enter "EXAMPLE".
- Select **Sections under pressure** from the drop-down list.
- Select **Hydraulic calculations will take place** from the drop down list.

Select **Ok**.

## 2.3 Step 03: Data input

Select **Friction Calculation** from the **Data** menu:



The dialog box titled "Friction calculation algorithm..." contains the following settings:

- Turbulent flow**: Turbulent flow if Reynolds is greater than ; Friction type: Darcy - Swamee - Jain.
- Laminar flow**: Laminar flow if Reynolds is lower than ; Friction type: Darcy - Laminar.
- Transient flow**: Friction type: Darcy - Moody Cubic.
- Settings**: Accuracy: ; Maximum number of trials: ; Averaging formula: Pavlovskii.
- ☐ Set as defaults for all new projects.
- Buttons: Ok, Cancel.

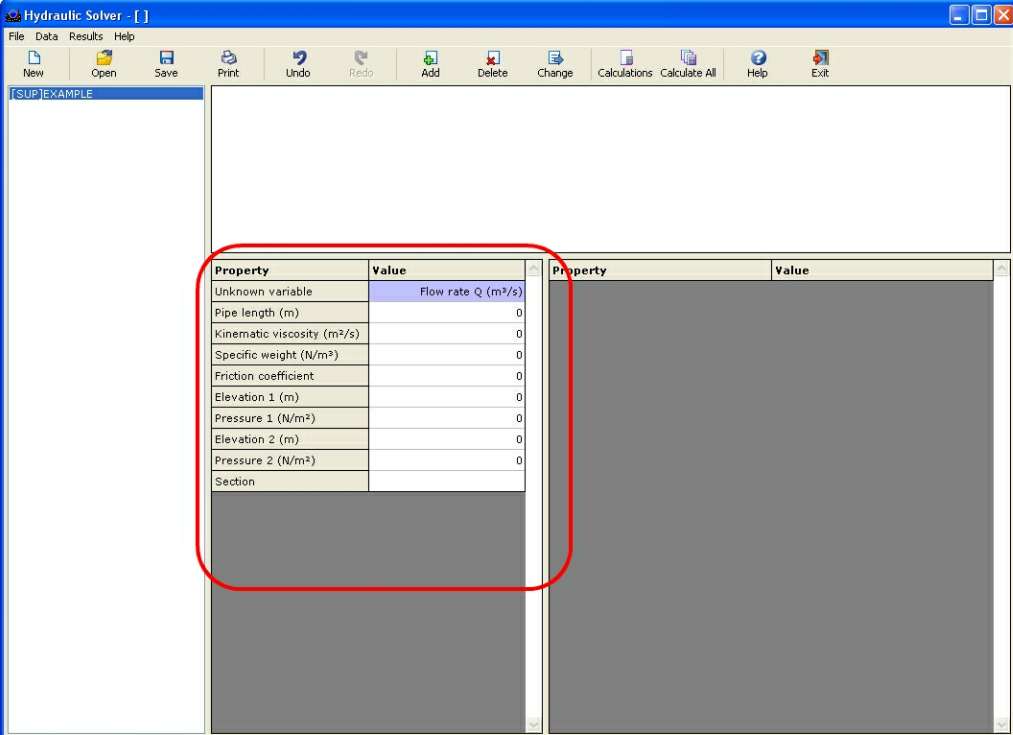
Enter the following data:

In **Turbulent flow** type "**4000**" in the **Turbulent flow if Reynolds is greater than** field and select **Darcy - Swamee - Jain** as the **Friction type**.

In **Laminar flow** type "**4000**" in the **Laminar flow if Reynolds is lower than** field and select **Darcy - Laminar** as the **Friction type**.

In **Transient flow** select **Darcy - Moody Cubic** as the **Friction type** and press **Ok**.

Enter the rest of the data by typing directly into the left grid of the main form:

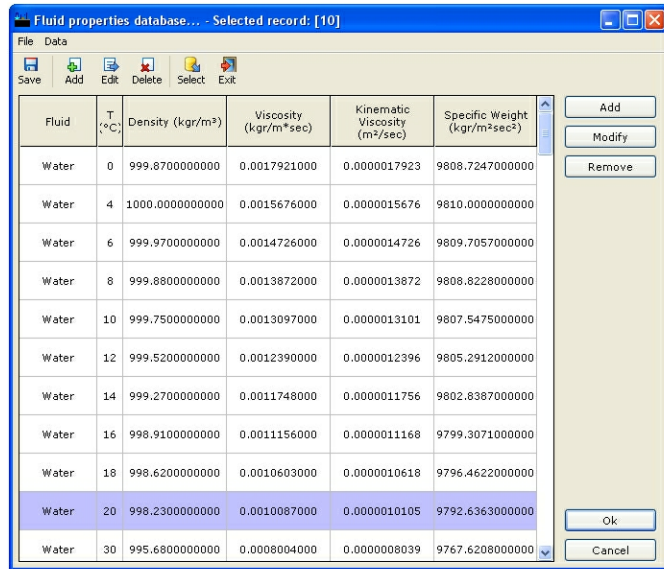


The main form of the Hydraulic Solver software shows a grid for data input. The left grid is highlighted with a red rectangle and contains the following data:

Property	Value
Unknown variable	Flow rate Q (m³/s)
Pipe length (m)	0
Kinematic viscosity (m²/s)	0
Specific weight (N/m³)	0
Friction coefficient	0
Elevation 1 (m)	0
Pressure 1 (N/m²)	0
Elevation 2 (m)	0
Pressure 2 (N/m²)	0
Section	

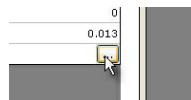
Enter the following data:

- **Unknown variable:** Select **Flow rate Q ( $\text{m}^3/\text{s}$ )**.
- **Pipe length:** type "200" and hit ENTER. Note that the unit conversion button is displayed, with which you can enter the pipe length in another unit system.
- **Kinematic viscosity:** Click on the cell, select the button with the ellipses (...) to invoke the corresponding database, select water 20°C and press Ok:



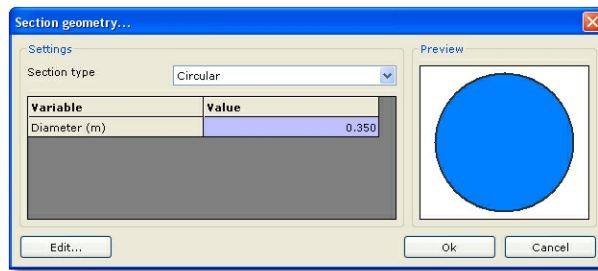
Fluid	T (°C)	Density (kg/m³)	Viscosity (kg/m·sec)	Kinematic Viscosity (m²/sec)	Specific Weight (kg/m²·sec²)
Water	0	999.8700000000	0.0017921000	0.0000017923	9808.7247000000
Water	4	1000.0000000000	0.0015676000	0.0000015676	9810.0000000000
Water	6	999.9700000000	0.0014726000	0.0000014726	9809.7057000000
Water	8	999.8800000000	0.0013872000	0.0000013872	9808.8228000000
Water	10	999.7500000000	0.0013097000	0.0000013101	9807.5475000000
Water	12	999.5200000000	0.0012390000	0.0000012396	9805.2912000000
Water	14	999.2700000000	0.0011748000	0.0000011756	9802.8387000000
Water	16	998.9100000000	0.0011156000	0.0000011168	9799.3071000000
Water	18	998.6200000000	0.0010603000	0.0000010618	9796.4622000000
Water	20	998.2300000000	0.0010087000	0.0000010105	9792.6363000000
Water	30	995.6800000000	0.0008004000	0.0000008039	9767.6208000000

- **Specific weight:** Click on the cell, select the button with the ellipses (...) to invoke the corresponding database, select water 20°C and press Ok.
- **Friction coefficient:** type "0.02" and hit ENTER.
- **Elevation 1:** The pipe slope is zero therefore you can ignore this field
- **Pressure 1:** The program expects all values to be in  $\text{N/m}^2$  so click on the cell and the click on the **U** button that shows up to invoke the **Unit conversion** routine. In **Unit conversion** select **Atmospheres** as the unit, type "20" in the next field and click **Ok**.
- **Elevation 1:** The pipe slope is zero therefore you can ignore this field
- **Pressure 1:** Type "0" and press ENTER.
- **Section:** A button with ellipses (...) appears when you double-click into the cell:



Select this button to invoke the section type form:





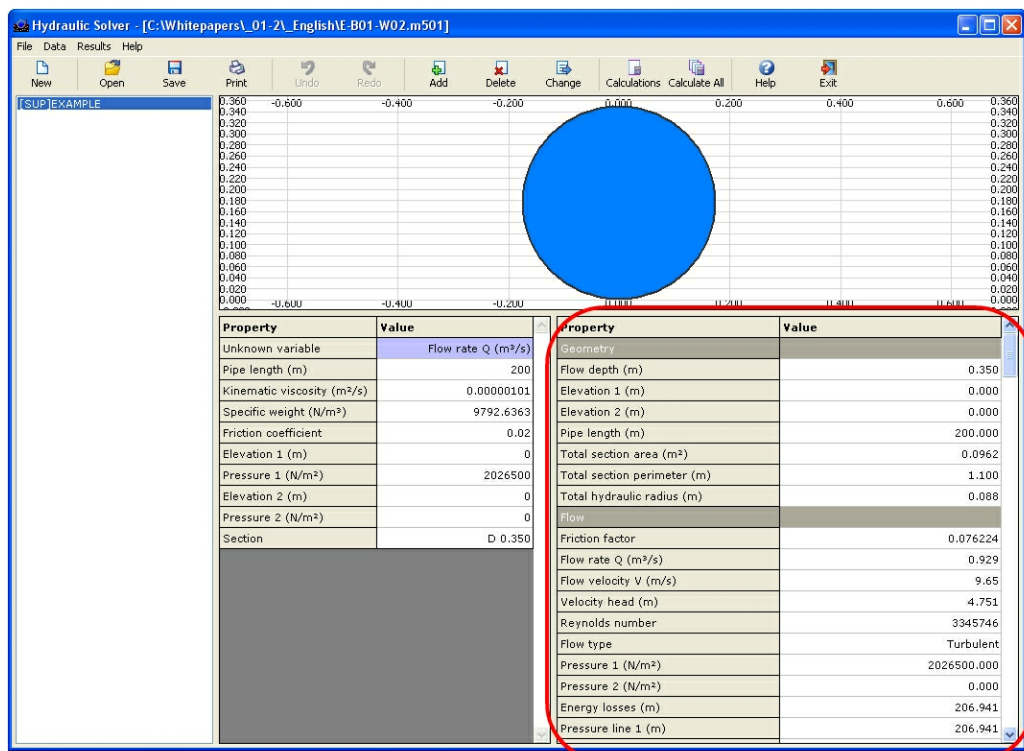
Enter the following data:

- **Section type:** Select **Circular** from the drop-down list.
- **Width (m):** type "0.350" and hit ENTER.

The program will display a preview of the section when it is 100% full. Select **Ok**.

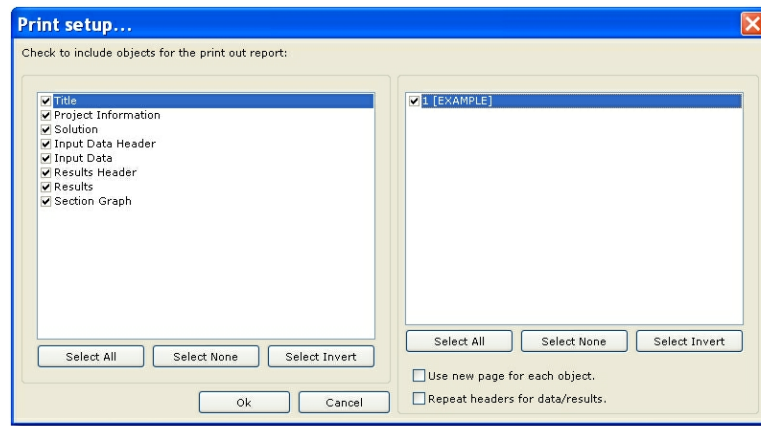
## 2.4 Step 04: Calculations

The calculations are performed automatically. The results are displayed in the right grid of the main form:



## 2.5 Step 05: Printing

Optionally, you may want to print the results to a printer, to Microsoft Word or Microsoft Excel. From the **File** menu select **Print setup**:



Select the solution "EXAMPLE" from the list on the right and select **Ok**.

From the **File** menu select **Print** or **Print To > Word** or **Print To > Excel** to invoke the corresponding print engine.

# Chapter

---



## 3 Help

### 3.1 Technical support

#### **Technical Support**

TechnoLogismiki offers technical support 24 hours per day, 365 days per year, through the Web site where you can get information on the latest programs and services.

#### **Support by e-mail**

Please use the dedicated e-mail addresses for better customer service:

- for questions regarding sales: [sales@technologismiki.com](mailto:sales@technologismiki.com)
- for questions regarding the usage of programs: [support@technologismiki.com](mailto:support@technologismiki.com)
- for any other question or comment: [info@technologismiki.com](mailto:info@technologismiki.com)

The normal response time is within two business days. If your inquiry cannot be answered via e-mail, a customer service representative will contact you via telephone.

#### **Interactive Support**

Business days, 09:00 - 17:00 Eastern European Time:

- Telephone [3 lines]: ++30-210-656-4147
- FAX: ++30-210-654-8461
- Address: 5, Imitou str, Cholargos, 15561, Athens, Greece.