Trajectory Plotting System

http://repository.unimelb.edu.au/10187/2415

Version 1.2

USERS' MANUAL

for use with

Windows - based Personal Computer

First Edition

R. H. Macmillan
Senior Fellow in Agricultural Engineering
International Development Technologies Centre
University of Melbourne
rhmacm@unimelb.edu.au

April 2007

Note: This is the only form of 'help'; there are no 'help screens' with the program.
CONTENTS

PREFACE

1.0 OVERVIEW

2.0 INTRODUCTION

2.1 PROGRAM REQUIREMENTS

2.2 INSTALLING THE PROGRAM

3.0 THE TPS PROGRAM

3.1 INTRODUCTION

3.2 PROPERTIES AND PARAMETERS

(a) Particle
(b) Fluid

3.3 END-LINE PARAMETERS

3.4 DEFAULT VALUES AND NAMES

(a) Particle
(b) Fluid
(c) End lines
(d) Screen
(e) Graph
(f) Time step
(g) Program run time
(h) Reynolds Number
(i) Layers

4.0 USING THE PROGRAM

4.1 GENERAL

4.2 CONSTANT FLUID SCREEN

(a) Insertion of data for the particle
(b) Insertion of data for the fluid
(c) Insertion of end-lines
(d) Calculating trajectories

4.3 VARIABLE FLUID SCREEN

4.3.1 Inserting layers

4.3.2 Inserting trajectory data

(a) Insertion of data for particle
(b) Insertion of data for fluid
(c) Insertion of layers
(d) Insertion of end-lines
(f) Calculating trajectories
4.4 VIEWING AND SAVING RESULTS

4.4.1 Results

(a) Default screen
(b) Other screens
(c) Values along the trajectories
(d) Chart title

4.4.2 Saving results

(a) Default screen
(b) Other screens
(c) Workings file

GLOSSARY OF TERMS

Symbols
Subscripts
Trajectory Plotting System

Preface

The computer program for plotting particle trajectories arose out of the author's experience in teaching and research in agricultural engineering at the University of Melbourne. Given the present widespread use of the digital computer it was considered desirable that a general program which will solve 'all' problems should be available. This Users' Manual is provided to assist in its use.

The program is based on the theory of a particle moving relative to a fluid under the action of gravity and fluid frictional forces only. It enables the user to specify conditions for a fluid and the initial conditions for particle, both moving in any direction. It plots the trajectories for up to 10 particles and gives the final values at a user defined end line.

Readers who are not familiar with fluid-particle mechanics may find the associated monograph helpful in giving the background theory and some worked examples; see below.

The program is a general one that has been written primarily with the needs of University engineering students and engineering designers in mind. Part I of the monograph has therefore been mainly directed at providing the general background theory. Parts II to IV provide examples with special reference to agriculture and associated technologies.

Engineers undertaking research studies in fields associated with fluid-particle mechanics may also find the program useful. It may also be applied to simple problems that arise in the teaching of physics and applied mathematics at a senior secondary school level.

The author wishes to acknowledge the work of students in Software Engineering in the Department of Computer Science, Faculty of Engineering at the University of Melbourne who developed the original version of the program for MacIntosh in 1995 and to AIMTEC who developed the Windows based version described in this manual.

The author also wishes to acknowledge the funding of the latter development by the University of Melbourne with a grant from the G.H and F. R. Vasey Bequest. The interest and support of Dr Graham Moore in arranging the contract is much appreciated.

The program is available without cost on the University of Melbourne web-site. However Universities and commercial organizations in the developed world who make significant use of the program are encouraged to make an appropriate donation to G.H and F. R. Vasey Bequest at the University of Melbourne. These could be sent, with an explanatory letter, to The Registrar, University of Melbourne, Parkville, Victoria 3052, Australia.

All who use the program are also encouraged to acknowledge such use. The author would also be interested to hear of this, also any errors or omissions.

For a detailed consideration of the background theory and a number of worked examples the reader is referred to the associated monograph entitled:

The Mechanics of Fluid - Particle Systems
http://repository.unimelb.edu.au/10187/440

This illustrates the use of the program in processes associated with agriculture and agricultural engineering and gives a small number of examples in other fields.

R.H. Macmillan
rhmacm@unimelb.edu.au

April 2007

Photo credit: The use in the opening screen of photographs from the following is gratefully acknowledged:

- Vicon / Kverneland Group
- World Food Program and F. Caracciolo
- John Deere Australia

1 Except ballistics, atomic particles etc.
1.0 OVERVIEW

The 'trajectory plotting system' described in this Manual allows the user to analyse any processes involving relative motion between a particle and a fluid in a gravity field and plot the results of up to 10 analyses on one screen.

The program is based on a planar rectangular co-ordinate system in which the horizontal and vertical components of the motion are plotted in the x and y directions, increasing positive to the right and upwards respectively. This analysis may only be done in two dimensions, i.e., in one plane; three dimensional analyses are not possible.

The analysis is based on the relative movement of a particle in a fluid under the action of two forces viz., gravity (corrected for buoyancy) and fluid drag. The latter is calculated according to well known fluid mechanics based on the drag coefficient - Reynolds number relationship. Other forces such as those due to electrostatic charge or particle spin are not considered. The program also assumes that the mass of the particle is constant.

Non spherical particles can be analysed if their drag coefficient - Reynolds number relationship is known. However, because such particles will probably spin and swerve and also accelerate and decelerate, the predicted motion will be the gross or macro value; the instantaneous motion at the micro level cannot be determined.

The program also assumes that there is only one particle and that there are no effects from other particles, as frequently occurs in practical situations. The effect of multiple particles is discussed briefly in Section 2.9 of the associated monograph.

The program is general in the sense that it will predict the motion of a particle dropped or projected in any direction in a fluid moving in any direction. The fluid is assumed to be moving at a constant velocity except in situations where it is possible to model the fluid stream as a series of up to 10 parallel layers with a constant velocity in each. Hence a boundary layer can be simulated.

The program is based on Newton's Second Law, viz:

\[
\text{Resultant force} = \text{mass of particle} \times \text{acceleration of the particle}
\]

The resultant force is given by the numerical vector addition of the gravity force and the fluid drag force which in turn is based on the velocity of the particle relative to the fluid. The double numerical integration of the acceleration gives the horizontal (x) and vertical (y) distances. These are shown plotted against each other as well as against the time period.

Up to ten graphs can be shown on one page. They can either be for:

* Ten different particles as represented by their different properties (diameters, masses) or parameters (initial velocities including magnitude and angle)\(^2\) for one fluid condition.

* Ten different fluid velocities (magnitude and angle)\(^2\) in layers for up to 10 particle conditions (as above). It is not possible to have different fluid properties (i.e., density and viscosity), i.e., have different fluid types. Where it is desired to analyse a particle moving between two fluids the output variables for the first can be used as the input to the second.

The program analyses and plots within end-lines which the user specifies to define the area of interest. The magnitude of the time steps over which the calculations are done can also be specified by the user.

Values at the end-line are reported as the final values of x, y, velocity, angle and the time elapsed.

\(^2\) For convenience in the following, 'velocity magnitude' will be shortened to 'velocity' and 'velocity angle' to angle.
2.0 INTRODUCTION

2.1 PROGRAM REQUIREMENTS

* Pentium Based Central Processing Unit (CPU)
* 16MB RAM
* 5MB Free HDD space
* Color printer (Optional)

2.2 INSTALLING THE PROGRAM

Run the Trajectory Plotting System - Install.exe file and follow the on-screen prompts.

3.0 THE TPS PROGRAM

3.1 INTRODUCTION

The use of the program is based around two main screens according to which are the constant factors for the fluid and which are the variable factors. For both screens the particle properties and parameters can be varied. The screens may be described as:

(i) constant fluid properties and parameters (the initial default screen).
(ii) constant fluid properties and at a constant angle but with a series of layers of chosen velocity and thickness or height. This screen is accessed by checking the 'use layers' box.

These will be termed the 'constant fluid' and the 'variable fluid' screens respectively. For both screens, the particle values are specified on the lower left of the screens and the fluid values on the lower right. The end lines appear in the lower centre and the results for the chosen trajectory at the specified end-line appear at the upper right. The graph in the upper left and centre of the screen will be x-y (the default condition) or x-t or y-t according to the type chosen in the upper horizontal menu; see Figures 2 and 3.

3.2 PROPERTIES\(^3\) AND PARAMETERS\(^4\)

The conditions or variables defining the analysis are as follows.\(^5\)

3.2.1 Particle

(a) Properties
* mass (g - gram) \([m]\)
* diameter (mm - millimetre) \([d]\)

(b) Parameters
* height (m – metre, the initial position of the particle on the y axis at x=0 relative to the user defined position at y=0)
* initial velocity (m/s - metre/second) \([U]\)
* initial angle (deg - degree, anti-clockwise from positive x axis) \([\phi]\)

---

\(^3\) The term ‘properties’ is used to refer to the inherent defining characteristics of the fluid and the particle, as shown.
\(^4\) The term ‘parameters’ is used to refer to the values specified by the user to define the (dynamic) conditions of the fluid and the particle, as shown.
\(^5\) The symbols shown in square brackets \([\ ]\) do not appear on the screen. They refer to the analysis given in the associated monograph ‘The Mechanics of Fluid - Particle Systems’.
3.2.2 Fluid

(a) Properties
* density (kg/m\(^3\) - kilogram/cubic metre) \([\rho]\)
* viscosity (mPa.s - milli Pascal.second = milli Newton/square metre .second) \([\mu]\)

(b) Parameters
* velocity (m/s, metre per second) \([V]\)
* angle (deg – degree, anti-clockwise from positive x axis) \([\theta]\)

Normally the data entered for the calculations as used in this program would be in a consistent set of units, eg, the International System, ie, metre, kilogram and second (MKS). However for convenience, constants within the program allow the use of the (inconsistent) units shown above and convert the data to the MKS system.

3.3 End-line Parameters

The program requires the user to specify an 'end-line' or 'lines' according to the area of interest. The default method is to specify an 'infinite' line by an x-y co-ordinate and an angle measured anti-clockwise from the positive x direction. If more than one line is required the 'bound' box will enable the user to specify a series of end-lines by means of their (x,y) co-ordinates. These might for example coincide with some or all of the boundaries of a machine.

For example, consider a particle injected horizontally onto a vertical wind tunnel with air passing upward to the atmosphere, Figure 1

The upper end-line would be the upper extent of the tunnel and the lower end-line would be the lower extent. A further end-line might coincide with the wall of the tunnel. A fourth end-line might also be used on the second tunnel wall but this would be unnecessary as no particles would reach this position.

Figure 1 Example of location of end-lines
3.4 DEFAULT VALUES AND NAMES

(a) Particle

(i) Default name - Particle 1

(ii) Default properties

* Mass – 1 gram
* Diameter – 1 mm

(iii) Default initial parameters

* Height – 1 m; the release point of the particle is at y=1.
* Velocity – 1 m/s; the particle has a velocity of 1 m/s.
* Angle – 0 degrees; the particle has a velocity in the direction of positive x axis.

(b) Fluid

(i) Default name - Air

(ii) Default properties

* Density - 1.2 kg/m³; the density of air at 20°C.
* Viscosity - 0.018 mPa.s; the absolute viscosity of air at 20°C.

(iii) Default parameters

* Velocity – 0 m/s; the particle moves in still air.
* Angle – 0 degrees; if the fluid has a velocity it is in the direction of the positive x axis.

(c) End - lines

(i) Default y intercept = 0 ; the end-line passes through the origin (x=0, y=0)

(ii) Default angle = 0 ; the end-line is parallel with the x axis

(d) Screen

Default arrangement: Constant fluid conditions with up to ten particle conditions.

(e) Graph

Default graph: x, y

(f) Time step

Default time step: 0.001 (1 milli-second)

The program uses a stepwise procedure to integrate the equations of motion and plot the resulting trajectories. The magnitude of the time step can be chosen by the user in the drop down menu under 'Scenario' at the top left of the screen. The usual value is in the range 1 to 100 millisecond. Decreasing the time step will, within reason, increase the accuracy of the calculation.

(g) Program run time

Default maximum run time : 50 seconds

The program will stop after an equivalent trajectory time of 50 seconds if an end-line has not been encountered.

1 These cells must have values greater than zero.
2 These cells have non zero values to illustrate the typical trajectory on the default screen.
(h) Reynolds number

The theory for the TPS program, which uses a basic fluid mechanics of a particle moving relative to a fluid, involves the non-dimensional parameter Reynolds number. This is used to correlate with the similarly non-dimensional drag coefficient \( C \) which in turn is used to calculate the drag force \( D \).

In calculating the Reynolds number the assumption is usually made that the particle is spherical and the well-known drag coefficient - Reynolds number relationship for spheres is used. This is the default relationship that is used in the program. The associated data can be seen under the 'scenario' drop down menu at the top of the screen.

If an alternative drag coefficient - Reynolds number relationship is known, this can be inserted in the program at that point and kept, under an appropriate name, for future use.

On the Reynolds number screen the following meanings apply:

* Clear - removes the existing visible data set (but not the default values)
* Insert - allows the user to insert new C and R values to give a new data set
* Remove - allows user to remove individual C and R values from the data set
* Save - allows the user to save a data set under a relevant name to be specified
* Load - opens a menu to allow the user to choose a stored relationship
* Sort - sorts the data set
* Use defaults - loads the default data set (for spheres)

When a scenario (screen with trajectories, parameters, properties, etc) is saved, the drag coefficient - Reynolds number data set which was used to calculate it is also saved. When that scenario is later opened the relevant data set will also be loaded.

When a 'new' screen is opened the default values for spheres will be loaded.

Further details regarding the drag coefficient - Reynolds number relationship and its use in the program can be found in the author's monograph:

''The Mechanics of Fluid –Particle Systems'.
http://eprints.unimelb.edu.au/archive/00001514/

(i) Layers

Layers are used to specify a variable velocity regime through which the particle must pass as discussed in Section 4.3. Up to 10 different layers with user specified heights and velocities can be used.

In this context, the following default values are given:

* Y intercept refers to the intercept on the y axis of the lowest edge of the lowest layer; the default value is zero.

* Angle refers to the (constant) angle (with respect to the positive x axis) of the set of layers; the default value is zero.

* Velocity refers to the user defined velocity in each layer; the default values are zero.

* Height refers to the user defined (vertical) height of each layer. To avoid problems with the calculations (in the default screen) the default value of the height for 'Layer 1' is specified as 1 metre. This value will not be significant until the velocity in this layer is changed from the default value of zero.
Figure 2: The default screen – constant fluid

Figure 3: The default screen – variable fluid
4.0 USING THE TPS PROGRAM

4.1 GENERAL

As noted above, two types of problem can be solved with the trajectory plotting system.

In both, the particle conditions (properties and parameters) can be varied and trajectories plotted on a single screen for:

(i) constant fluid conditions (properties and parameters) at any angle. The default screen applies to this type of problem.

(ii) variable fluid conditions (parameters only) where a series of up to 10 parallel layers of chosen velocity and height can be specified at any angle less than 90 degrees. The 'use layers' box is checked in this type of problem.

The features of the program are illustrated by the respective screens.

4.2 CONSTANT FLUID SCREEN

The screen will appear as shown in Figure 1. The insertion point will move if the tab key is used.

(a) Insertion of data for the particle

(i) Insert the name of the first particle and its trajectory; the default name is Particle 1. This should be chosen to define it in terms that will distinguish it from the other trajectories that will be plotted on the same screen.

Example: Imagine that it is desired to plot trajectories for two particles with
* a constant diameter (d)
* different masses (m₁, m₂)
projected
* at a constant angle (ϕ)
* with different velocities (u₁, u₂).

The 4 names could then be: m₁,u₁; m₁,u₂; m₂,u₁; m₂,u₂. The constant parameters and the meaning of variables would be recorded in the caption for the screen or graph.

(ii) Insert properties for the first particle

* mass of the particle (g); (the default value is 1)
* diameter of the particle, (mm); (the default value is 1)

(iii) Insert parameters (initial values) for the first particle

* height of release of particle with respect to assumed zero (m); (the default value is 1)
* velocity of particle (m/s); (the default value is 1)
* angle of the particle (deg) (the default value is 0)

(b) Insertion of data for the fluid

(i) Insert name of the fluid; the default name is 'Air'.

(ii) Insert properties for the fluid

* density of the fluid (kg/m³); (the default value is 1.2 for air at 20° C)
* viscosity of the fluid (mPa.s); (the default value is 0.018 for air at 20° C)

(iii) Insert parameters for fluid

* velocity (m/s); the default value is zero
* angle (deg.) ; the default value is zero
(c) Insertion of end-lines

These are set according to the area of interest on the (x, y) plane and the expected trajectories. The program will stop calculating the trajectories when any of these end-lines are crossed.

These can be:

(i) infinite - a single line which will extend as far as is needed. It is specified by:
   * an intercept (metre) on the y axis
   * an angle (degrees) specified anti-clockwise with respect to the positive x axis. The default value is the end-line passing through the point (0, 0) with zero angle.

(ii) bound – a series of lines each specified by two points. The number and position for these can be chosen according to the area of interest. There is no requirement for them to form an closed loop.

If an end-line is encountered the program will stop, plot the resultant trajectory and show the results on the 'Trajectory end results' area. If an end-line is not encountered the program will stop after the equivalent trajectory time specified in the Scenario drop down menu. The default time is 50 seconds.

End-lines can only be specified by x and y co-ordinates. Time cannot be used to define an end-line.

Note that it is not necessary to specify all of the end-lines at the beginning of an analysis. If after starting to plot a series of trajectories it is found that one or more does not reach an end-line then this can be altered so that it does so.

(d) Calculating the trajectory

To calculate the trajectory press the 'calculate' button.

Repeat steps (i) and (ii) under (a) above. Use the 'add' box in the lower left hand corner to insert another particle name and its appropriate data. Modify the end lines as appropriate.

4.3 VARIABLE FLUID SCREEN

4.3.1 Inserting layers

The variable fluid screen allows the use of a set of up to 10 fluid layers. The screen will appear as shown in Figure 3 when the 'Use layers' box is checked. All layers must be for the same fluid with the same properties. See Figure 4 for two typical layouts. Specify:

(i) angle of the set. The angle is specified in the 'angle' box and can be any value less than 90 degrees (anti-clockwise from the positive x axis).

(ii) position of the set relative to the chosen zero. This is specified by the 'y intercept' box. The bottom of the lowest layer (layer 1) will be at this point. Other layers (2, 3, 4, etc) will be above this.

(iii) height of the layers. This is measured in the vertical direction and will be equal to the thickness of the layers when the angle of the set is zero.

(iv) velocity in each layer
4.3.2 Inserting trajectory data

(a) Insertion of data for the particle

(i) Insert the name of the first trajectory. This should be chosen to define it in terms that will distinguish it from the other trajectories that will be plotted on the same screen. (See example given in 4.2 (a) above.

(ii) Insert properties for the first particle:
* diameter of particle (mm)
* mass of the particle (g)

(iii) Insert parameters (initial values) for the first particle
* velocity of the particle (m/s)
* angle of the particle (deg)
* height of release of particle with respect to zero for the trajectories(m)
(b) Insertion of data for the fluid

(i) Insert name of the fluid; the default name is 'Air'

(ii) Insert properties of the fluid (the default values are for air at 20° C)
* density of the fluid (kg/m³)
* viscosity of the fluid (mPa.s)

(c) Insertion of the set of layers

(i) Insert the angle for the layers (it must be the same for each and less than 90 degrees)

(ii) Insert the y intercept for the bottom edge of the lowest layer. The lowest layer may of course have zero velocity if it is desired that the particle should pass through layers with a finite velocity into still fluid. (see under 'Layers')

(iii) Insert the parameters (height (m) and velocity (m/s)) of each layer. The layers will be inserted and numbered in sequence from the lowest to the highest. The height refers to the height of the layer in the vertical direction. Where used for descriptive purposes the term 'thickness' refers to the thickness of the layer perpendicular to the layer. Height and thickness will be equal when the angle of the set is zero.

(d) Insertion of end-lines

Insert the other end lines according to the area of interest and the expected trajectories. These may be at any angle and position.

(e) Calculating the trajectory

To calculate the trajectory press the 'calculate' button.

Repeat steps under (a) above. Use the 'add' box in the lower left hand corner to insert another particle name and its appropriate data. Modify the end lines as appropriate.
4.4 Viewing and Saving Results

4.4.1 Results

(a) The default screen

The default screen shows the results:

(i) as trajectories for all particles on the graph section of the screen. These can be shown in one of the following forms:

* x, y
* x, t
* y, t

The default condition is x, y. The trajectories of the individual particles can be identified by color in the legend to the right of the graph or by form of line when the line is printed in black.

(ii) the ‘trajectory end results’ for the particular particle as chosen in the drop down menu in the section in the upper right hand corner of the screen. The results show the following when the trajectory crosses an end line:

* x co-ordinate (m)
* y co-ordinate (m)
* time elapsed for the trajectory (sec)
* magnitude of the particle velocity (m/s)
* angle of the particle velocity measured anti-clockwise from the positive x axis (deg).

Where an end-line is specified, for example \( y_i = 0 \) as shown in Figure 2 the final y value may not be exactly zero. This is because the program stops calculating the trajectory only after the end-line has been crossed and shows the extent by which the final calculation exceeds the exact value at the end-line. The excess can be reduced by reducing the time step.

(b) Other screens

Two other screen formats can be used to view the results; choose under the ‘View’ drop down menu.

(i) With only the ‘Results’ box checked the screen shows all the trajectories (enlarged vertically) together with the legend. The ‘trajectory end results’ shows the results for the particular particle as chosen in the drop down menu in the section in the upper right-hand corner of the screen.

(ii) With only the ‘Properties’ box checked the screen shows all the input data and trajectories (enlarged horizontally) with legend but not the individual results.

(c) Values along trajectory

The (x, y, t) values along the trajectories will be shown in the bottom right corner if the cursor is moved along the trajectory line.

(d) Chart title

A chart title appropriate to the screen can be specified in the box which will appear when the space above the graph is checked.

(e) Gridlines

Gridlines on the graph can be hidden and the spacing can be changed under the ‘Settings’ drop down menu.
4.4.2 Saving results

(a) Default screen

'Properties' and 'Results'

With 'Properties' and 'Results' boxes checked under 'View' menu the default screen shows the list of names of all the particles and their associated trajectories and legend plus the fluid data and end-lines.

However it only shows:
* the properties and initial parameters for the selected 'particle' in the 'Particles' area of the screen (lower left hand corner)
* the end results for selected 'particle' in the 'End results' area of the screen (upper right hand corner)

When printed this screen shows all the trajectories together with the fluid data and the names, data and end results for all the particles. This can either be portrait or landscape on A4 sheet.

Note, the default screens can only be printed as they appear by means of the in-built 'print screen' command on the computer.

(b) Other screens (see 4.4.1 (b) above)

(i) With the 'properties' box only checked under the 'View' menu the screen is as in (a) above but without the results section. When printed the particle and fluid data and results are listed underneath.

(ii) With the 'results' box only checked under the 'View' menu the screen is as in (a) above but without the properties section. When printed the particle and fluid data and results are listed underneath.

(c) Printing results

The above results can be printed 'landscape' or portrait' form.

To print the results to another application go to 'Copy graph image to clip board' in Edit menu. Color or black can be chosen but if the latter is used the form of the graph as shown in the legend will be used to identify the various trajectories.

This image can then be pasted to the application and be edited as desired

(e) Workings file

Certain results from the calculations (as specified within the program) for the individual time steps can be recorded if a 'workings' file is available in the folder containing the program. This file can be printed to an XL file for further examination or analysis. The number of data points will be determined by the time step chosen. Remove the workings file if the recording of the data is not desired.
### Glossary of Terms

The equation numbers refer to equations in the monograph: The Mechanics of Fluid - Particle Systems

http://eprints.unimelb.edu.au/archive/00001514/

**Angle of fluid velocity, \( \theta \)**

Angle of the fluid velocity (\( V \), relative to the **earth**) measured anti-clockwise from the positive horizontal \( x \)-axis; (degrees).

**Angle of particle velocity, \( \phi \)**

Angle of the particle velocity (\( U \), relative to the **earth**) measured anti-clockwise from the positive horizontal \( x \)-axis; (degrees).

**Angle of relative particle velocity, \( \alpha \)**

Angle of the particle velocity (\( W \), relative to the **fluid**) measured anti-clockwise from the positive horizontal axis (\( x \)-axis); (degrees).

**Area of particle, \( A \)**

The projected area of the particle: 

\[
A = \frac{\pi}{4} d^2
\]

**Co-ordinate system**

The dimensional system on which the trajectory is drawn. An \( x \)-\( y \) plot where \( x \) is the horizontal distance and \( y \) the vertical distance measured from the defined zero point (not necessarily the initial point for the particle).

**Density, \( \rho \)**

The mass per unit volume.

**Diameter, \( d \)**

The (mean, equivalent, effective or actual) diameter of the particle; (mm)

**Drag, \( D \)**

See fluid drag.

**Drag coefficient, \( C \)**

The dimensionless coefficient of proportionality in Newton's equation for fluid drag,

\[
D = C A_p \frac{\rho_f W_{pf}^2}{2}
\]

(Equation 2.2)

\( C \) is not a constant but is a function of the flow conditions around the particle and hence is usually plotted against the Reynolds number.

**End-line**

The line or frame that determines where the calculation of the trajectory will terminate.

**Fluid density, \( \rho_f \)**

Mass per unit volume of fluid, kilogram per cubic metre; (kg/m\(^3\))

**Fluid drag, \( D \)**

The force opposing the relative motion of the particle and fluid. It is given by Newton's equation for fluid drag and in this work is assumed to act in a direction opposite to the relative particle velocity.

\[
D \propto A_p \rho_f W_{pf}^2
\]

(Equation 2.1)

**Fluid layer**

Band of fluid at a user defined angle with a user defined velocity and height; not available if vertical.

**Fluid properties**

The values that specify the fluid, namely, the density and viscosity.
**Fluid velocity (magnitude),** $V$ or $V_{fe}$
Magnitude of the velocity of the fluid (relative to the earth) through which the particle passes; metre per second, (m/s).

**Fluid (velocity) angle,** $\theta$
Angle of velocity of fluid, measured anti-clockwise from the positive horizontal x-axis; (degrees).

**Fluid viscosity,** $\mu$
The property of a fluid whereby it resists relative deformation in shear. The absolute viscosity is the constant of proportionality between the velocity gradient and the shear stress; milliPascal.second, (mPa.s = mN.s/m²)

**Frame of reference**
The ‘fixed’ base against which motion is measured, usually the earth. However an alternative frame of reference (such as a machine) may itself be moving (with a constant velocity) on the earth.

**Grid lines**
Co-ordinate lines on the graphs coinciding with scale markers on their axes.

**Height**
(i) Initial distance of the particle above (+) or below (-) the origin of the co-ordinate system; metre; (m).
(ii) The height of a fluid layer measured in vertical direction; (m)

**Landscape**
Sideways orientation of pages.

**Layer**
A band of fluid with user defined constant height, angle and fluid velocity.

**Parameter**
A user defined numerical value specifying the dimensional arrangement and dynamic conditions for the fluid or particle.

**Particle density**
The mass per unit volume of a particle; gram per cubic mm (g/mm³).

**Particle velocity (magnitude),** $U$ or $U_{pe}$
The magnitude of the velocity of the particle (relative to the earth); metre per second, (m/s).

**Portrait**
Normal orientation for printed pages for text.

**Property**
An inherent or defining numerical value for a fluid or particle.

**Range**
The horizontal distance moved by the particle; metre (m)

**Relative particle velocity (magnitude),** $W$ or $W_{pf}$
The magnitude of the velocity of the particle (relative to the fluid); metre per second, (m/s).

**Reynolds number,** $R$
A dimensional number that is used to characterize the flow conditions around a particle.

$$ R = \frac{\rho_f W_{pf} d}{\mu_f} \quad \text{(Equation 2.3)} $$

**Time step**
The size of the time increment between calculated points on the trajectories. Specified by the user; default value 0.001 second.

**TPS**
The Trajectory Plotting System
**Trajectory**
The path taken by a particle in a fluid in a gravitational field as viewed on the x-y co-ordinate system usually with reference to the stationary earth. The trajectory can also be viewed as either an x-t or y-t plot.

**Trajectory list**
The list of the names given to the individual trajectories on a particular screen/page.

**Viscosity, \( \mu \)**
See fluid viscosity

**x-y graph**
A graph of the vertical (y, metre) position(s) of the particle(s), versus the horizontal (x, metre) position(s).

**x-t graph**
A graph of the horizontal (x, metre) position(s) of the particle(s), versus time (t, second).

**y-t graph**
A graph of the vertical (y, metre) position(s) of the particle(s), versus time (t, second).

**Symbols**

\[
\begin{align*}
\mu & = \text{Viscosity of fluid, mPa.s} \\
\rho_p & = \text{Density of particle, kg/m}^3 \\
\rho_f & = \text{Density of fluid, kg/m}^3 \\
d & = \text{Diameter of particle, mm} \\
m & = \text{Mass of particle, g} \\
A & = \text{Projected area of particle, mm}^2 \\
C & = \text{Drag coefficient corresponding to Reynolds number} \\
R & = \text{Reynolds number} \\
U & = \text{Velocity of particle relative to the earth (m/s)} \\
V & = \text{Velocity of fluid relative to the earth (m/s)} \\
W & = \text{Velocity of particle relative to the fluid (m/s)} \\
\Delta & = \text{Differential} \\
t & = \text{Time}
\end{align*}
\]

**Subscripts**

- \( f \) Fluid
- \( e \) 'Stationary' earth or other frame of reference if moving
- \( v \) Vertical
- \( h \) Horizontal
- \( pf \) Particle relative to the fluid
- \( pe \) Particle relative to the earth
- \( t \) Condition at terminal velocity
Minerva Access is the Institutional Repository of The University of Melbourne

Author/s:
Macmillan, R. H.

Title:
Trajectory Plotting System and Users Manual for use with Windows based Personal Computer

Date:
2007

Citation:

Publication Status:
Unpublished

Persistent Link:
http://hdl.handle.net/11343/34981

File Description:
Users manual

Terms and Conditions:
Terms and Conditions: Copyright in works deposited in Minerva Access is retained by the copyright owner. The work may not be altered without permission from the copyright owner. Readers may only download, print and save electronic copies of whole works for their own personal non-commercial use. Any use that exceeds these limits requires permission from the copyright owner. Attribution is essential when quoting or paraphrasing from these works.