Propulsion of the Putt-Putt Boat – I

V Sharadha and Jaywant H Arakeri

V Sharadha is a III year mechanical engineering student at Indian Institute of Technology, Chennai. This work was done when she was JNCASR summer fellow in 2003 at IISc.

Jaywant H Arakeri is Professor at Mechanical Engineering Department and in the Center for Product Development and Manufacture at Indian Institute of Science, Bangalore. His research is mainly in instability and the turbulence in fluid flows.

Keywords
Fluid dynamics, thermodynamics.

Introduction

The putt-putt boat is a little toy that has fascinated everyone from children to physicists for over a hundred years now. This boat is powered, not by a conventional heat engine, but by a motor that can be termed a pulsating water jet engine.

The propulsion mechanism that drives the putt-putt boat uses simple physical principles in an ingenious way. It basically consists of a small shallow chamber, covered by a thin copper diaphragm. This is connected to the water astern of the boat by two pipes of 3mm bore (See *Figure* 1). The two pipes are initially primed with water and the base of the chamber heated, using a candle. The diaphragm begins to vibrate and the boat moves forward making the characteristic putt-putt sound. The boat works as long as the flame glows.

Experiments and Observations

The aim of the experiments was to answer questions that immediately comes to one's mind: what causes the putt-putt sound, and is it somehow responsible for the propulsion? Are two pipes necessary for the boat operation?

Set-up

The apparatus basically consists of a small glass tank. For flow visualization, a high-speed camera (Kodak Motion Corder Analyzer) was run at (See *Box* 1) of 500 frames per second(fps)/1000fps. Both hydrogen bubbles and silica particles (100 microns) were used as tracers.

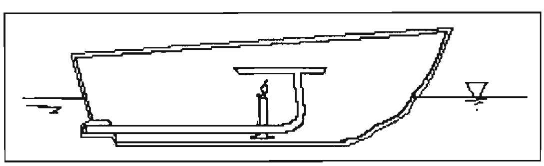


Figure 1. Sketch of the putt-putt boat.

Box 1. Experimental Techniques Employed

Hydrogen-Bubble Visualization

Flow visualization using hydrogen bubbles is a simple and popular method to study flow patterns.

Hydrogen bubbles produced by the electrolysis of water are used as tracer particles. Hydrogen is liberated at the cathode and oxygen at the anode. The quantity of the bubbles produced depends on the ion concentration in the water. Hence, electrolytes such as sodium chloride or sodium sulphate are added. The cathode (in our case a thin metal wire) is placed at the region of interest and the flow patterns are studied.

Particle Image Velocimetry (PIV)

This is a relatively newer technique of measuring velocities in a flow. The flow is seeded with neutrally buoyant particles which indicate the underlying fluid motion. The flow is recorded as successive frames on a camera at known time intervals. Corresponding flow regions in the successive frames are correlated to obtain the velocities.

A low-friction pulley system, a microphone and a larger tank were used respectively for thrust, frequency and the steady-state boat velocity measurements.

Observations

We can usefully think of the boat operation as consisting of two phases, suction and exhaust. During suction water from the outside enters the pipes; during the exhaust phase an equal amount of water is thrown out. We will see in the next part how a constant source of heat produces this periodic cycle of ejection and suction.

Improved flow visualization patterns were obtained with hydrogen bubbles as compared with the silica particles used in PIV.

The jets formed are approximately one centimeter below the free surface of the water. The two jets are in phase. The frequency of ejection obtained was 16 Hz, with a jet exit velocity of 41cm/s. The average steady-state velocity of the boat was approximately 20cm/s.

The boat operation consists of two phases, suction and exhaust.

An average weight of 600 mg was found to balance the mean thrust produced by the boat. Addition or reduction of 20 mg

The putt-putt sound seems to originate from the vibratory motion of the diaphragm.

resulted in the loss of equilibrium. This can be attributed to the change in the direction of the frictional force in the pulley, which is therefore less than 19.6 dynes (0.19mN), and this is small compared to the thrust measured. The thrust is thus about 6 milli-Newtons (mN).

The putt-putt sound seems to originate from the vibratory motion of the diaphragm, as it changes its configuration from concavity to convexity and back (ie., during suction and ejection strokes respectively). (See *Box* 2.) The diaphragm was thickened to reduce its amplitude of vibration and to observe its effect on the boat's motion. M-seal was used to restrict its vibration. The velocities of the unmodified and modified boats were found to be around 20 cm/s. Thus the thickening of the diaphragm

Box 2. Boat Frequency Measurements

The diaphragm is the source of the putt-putt sound, as the sound disappears when the diaphragm is thickened using m-seal. The sound of the boat was recorded onto a computer and the frequency of the putt-putting was measured to be 16Hz on an average. That is to say, the suction and ejection strokes take place 16 times a second. Each putt is like an impulise loading on the diaphagram and produced high frequency oscillations in the sound which is what we hear. Video recordings of the diaphragm were performed later to validate the frequency, measured from the microphone output.

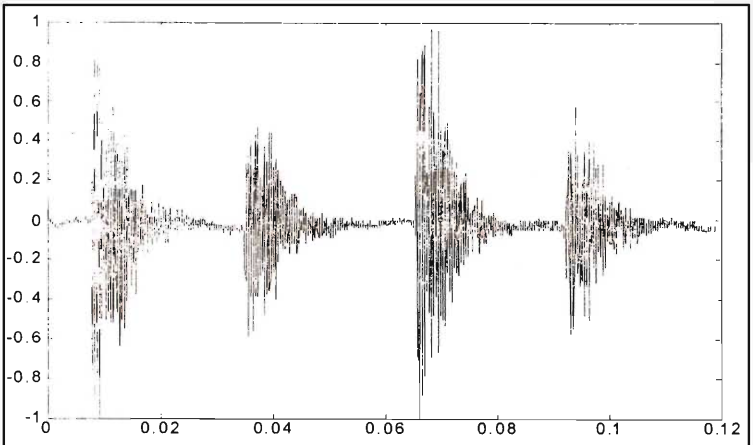


Figure A. Plot of normalized amplitude of the sound recorded, vs. time.

Figure A shows a plot of two cycles of the 'putt-putt' sound recorded. One cycle of the boat's operation can be thought to be composed of an ejection and suction. Thus, the two bursts of pulses in a cycle that we see in this figure can be thought of as corresponding to the outflow and inflow phases.

resulted in a perfectly operable, though noiseless putt-putt boat. The jet's motion seems unaffected by the diaphragm's presence as the boat works just the same with an inflexible boiler top. As to whether the motion of the diaphragm and the jet are in phase is yet to be determined conclusively.

Thickening of the diaphragm resulted in a perfectly operable, though noiseless putt-putt boat.

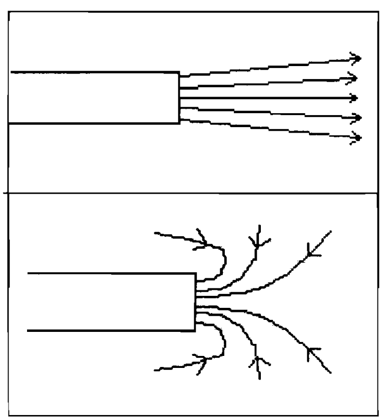
One of the pipes was plugged using a paper-wrapped matchstick and the boat was tested. The thrust produced was small and eventually the boat couldn't sustain operation. To observe the effect of the length of the pipes, a pair of 3cm long PVC tubes was used to lengthen them. The diaphragm frequency in this case was reduced to 13Hz approximately and the boat velocity also reduced to about 11 cm/s.

Principle of Operation

The boat is propelled forward by the alternate suction and ejection of the fluid across its pipes' orifice. The net mass flux is zero. Yet, there is a net forward thrust developed because, during the outflow, the flow is an axi-symmetric jet confined to a narrow domain and does not diverge much for reasonable distances, as in the exhaust from an automobile (*Figure* 2).

The nominally axi-symmetric flow during suction can be thought of as similar to the flow induced by a sink that is coincident with the pipe exit (*Figure* 3). The inflow of the same magnitude does not produce an equal and opposite thrust, and is small compared to the thrust produced during exhaust.

Figure 2 (top). During exhaust.
Figure 3 (bottom). During suction.

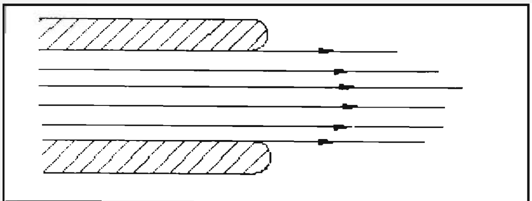


The difference between inflow and outflow is important in the study of fluid mechanics. It can be easily seen that we can easily 'blow out' a candle but cannot 'suck out' a candle even though the average velocity near the lips is nearly the same in each case (see *Box* 3).

Analysis

The analysis is basically carried out in two stages. In the following fluid mechanics section, we will analyze the forces that are instrumental in driving the boat. When the boat is

Box 3. Inflow and Outflow



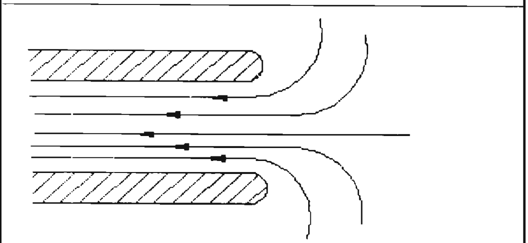


Figure B. (top) Out flow. Flow comes out like a jet.
Figure C. (bottom) Flow enters from all around.

The difference between inflow and outflow is an important one in the analysis of a fluid flow problems. During outflow the fluid comes out straight as a jet – a familiar example is a smoke laden exhaust from an automobile. As a consequence the pressure P_e across the jet near the exit is the atmospheric pressure P_a.(Further downstream ambient fluid gets dragged by the jet fluid and mixes with it and the jet grows.) Clearly Bernoulli's equation which would give

$$P_e = P_a - 1/2\rho U_e^2$$

is not applicable; U_e is the exit velocity.

During inflow fluid particles are drawn in from all directions. In this case, Bernoulli's equation is valid

$$P_I + 1/2\rho U_I^2 = P + 1/2\rho U^2 .$$

The pressure far enough away from the inlet is P_a and the velocity is zero. Thus pressure at the inlet

$$P_I = P_a - 1/2\rho U_I^2 ,$$

where U_I is the velocity at the inlet.

Viscosity is responsible for the difference between the inflow and the outflow flow patterns. For an ideal fluid (viscosity = 0) the flow patterns in both cases would be alike: during outflow the flow would be like the one shown in *Figure* C but with the flow directions reversed. For the outflow case the pressure would increase from a minimum value at the exit to the ambient value. In particular the flow next to the surface will experience an increasing pressure. Consider a small piece of a fluid flowing next to the surface (*Figure* D). This particle is finding increasing pressure values on its path and slowing down: at the same time fluid friction is also opposing the motion. The increasing pressure in the flow direction is known as adverse pressure gradient. The fluid particle cannot overcome this double barrier of pressure and friction.

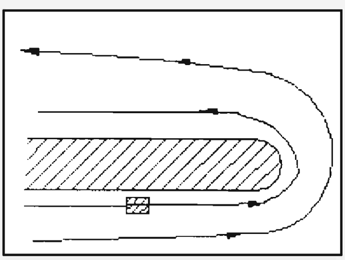


Figure D. A fluid particle trying to go around a corner will be opposed by viscous friction and pressure. Thus the flow will not be able to go around the corner, but go straight out as in Figure B.

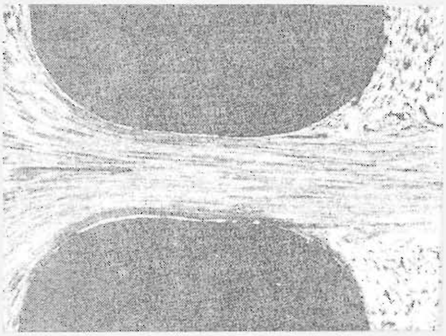
Box 3. continued...

Box 3. continued...

Any fluid however small its viscosity flowing past a surface in the presence of an adverse pressure gradient will separate. In the case of inflow, pressure is reducing in the flow direction (favorable pressure gradient) and flow does not separate; pressure is now aiding the motion. *Figure* E dramatically shows the difference between inflow and outflow.

This phenomenon of separation was first explained by Prandtl. Separation is in fact needed to explain the high drag forces on 'bluff bodies'. (See [4])

One consequence of the difference between inflow and outflow: we can easily blow out a candle flame but not suck it out.Try it ! Also, because of this difference we do not inhale the same air that was exhaled in the previous breathing cycle.



The above discussion is valid for low viscosity fluids (like water), or more correctly for high Reynolds number flows. For highly viscous flow the physics is quite different.

Figure E. Streamlines of flow through a channel which first converges and then diverges.

moving with a constant velocity the thrust averaged over one cycle is equal to the drag, caused by viscous stresses and the surface waves on the water.

Now, for ease of calculation, we consider inflow and outflow separately. The averages are still taken over one full cycle. During outflow, the jet being almost perpendicular to the orifice area, the pressure at the exit can be approximated to the ambient static pressure. It is during the outflow phase that almost all of the thrust is produced. But during inflow, the fluid will enter from all sides and the pressure at the tube inlet will be lower than the surrounding pressure. The thrust during inflow is likely to be small. See Classroom section (p.92) of this issue of *Resonance* for a detailed analysis.

Variable description:

Density	: ρ
Area of the exit plane	: a
Acceleration of the reference frame	: a_{rf}
Volume of the control volume	: V
Surface forces	: F_s
Body forces	: F_b
Velocity of the mass leaving the control volume	: U
Drag force	: D
Velocity of the boat	: U_∞
Jet velocity	: U_c

Box 4. Fluid Mechanical Analysis

Using the momentum conservation equation,

$$F_s + F_b - \int_V a_{rf}\, \rho\, dV = \int_{cs} U\, \rho U \cdot da + \frac{\partial}{\partial t}\left(\int_V U\, \rho\, dV\right) \cdot$$

After the boat has attained uniform velocity,
along the x-direction,

$$F_b = 0$$
$$F_s = \int \tau\, da.i = D$$
$$a_{rf} = 0$$

Upon averaging over a cycle of operation,

$$\frac{\partial}{\partial t}\left(\int_V U\, \rho\, dV\right) = 0 \cdot$$

Using the notation,

$$\frac{1}{T}\int_0^T G\, dt = \langle G \rangle,$$

the momentum conservation equation becomes,

$$\langle F_{sx} \rangle = \int_{cs} \langle U.i\, \rho\, U.da \rangle + \int_{cs} \langle p\, da.i \rangle.$$

During outflow, this equation becomes

$$\int_a \langle U_e.i\, \rho\, U_e.da \rangle + \int_a \langle p\, da.i \rangle\, \rho a \langle U_e \rangle^2.$$

During suction,

$$\int_a \langle U_i.i\, \rho\, U_e.da \rangle + \int_a \langle p\, da.i \rangle = 0.$$

Therefore the average drag force $\langle D \rangle \approx \rho a \langle U_e^2 \rangle = \langle thrust \rangle$, under steady-state conditions. Drag can also be given in terms of the drag coefficient C_D, as,

$D = C_D\, (\frac{1}{2} \rho U_x^2 A)$, where C_D can be obtained from correlations.

The thrust force on the boat is given by,

$$\langle \text{thrust} \rangle = \rho a \langle U^2_e \rangle \qquad \text{(see } \textit{Box} \text{ 4 and Classroom, p.92).}$$

The angular brackets indicate average over one cycle; ρ is the water density, a is jet area and U_e is jet velocity. The thrust is equal to the drag force.

From the velocity measured as 41 cm/s, the thrust calculations yield a value of 5.2mN for the thrust on the boat. This value compares well with the measured thrust values.

Modelling the boat as a flat plate, gives drag coefficient (CD) value of 0.0079, which yields a drag of 0.88 mN, when calculated using the wetted area. Using the values given for a kayak [3] and matching the Reynolds number of the toy boat and a kayak, we obtain a CD value of about 0.045 and a drag force of around 5.02mN which agrees quite well with the measured thrust value.

Conclusions

The putt-putt boat's operation has two phases, ejection and suction. During ejection water is thrown out of the pipes, and dur-ing suction an equal quantity of water is injested. Even though the net mass flow is zero, the differences in the flow patterns during outflow and inflow cause a net forward thrust; thrust is mainly produced during ejection phase.

Neglecting the suction part of the cycle for dynamic analysis of the boat agrees quite well with the experimental data obtained. Moreover, modeling the boat as a kayak, rather than a flat plate gives a better drag estimate. The diaphragm just makes the putt-putt sound and is not involved in the propulsion.

In the second part, the thermodynamic process involved in generating the thrust and drag forces will be dealt with.

Suggested Reading

[1] I Finnie and R L Curl, Physics in a toy boat, *American Journal of Physics*, Vol. 31, p. 289, 1962.

[2] J S Miller, Physics in a toy boat, *American Journal of Physics*, Vol. 26, p.199, 1958.

[3] Leo Lazauskas, John Winters, E O Tuck, Hydrodynamic drag of small sea kayaks, Courtesy: www.maths.adelaide.edu.au

[4] Jaywant H Arakeri and P N Shankar, Ludwig Prandtl and Boundary Layers in Fluid Flow, *Resonance*, Vol.5, No.12, pp.48-63, 2000.

Address for Correspondence
V Sharadha
III Year
Mechanical Engineering
Student
Indian Institute of Technology
Chennai 600 036
Jaywant H Arakeri
Mechanical Engineering
Department
Indian Institute of Science
Bangalore 560 012, India.
Email:
jaywant@mecheng.iisc.ernet.in