THE HYDRAULIC JUMP: A TEST CASE FOR THE SPH MODEL.

David López Gómez[1]

ABSTRACT

Mathematical models applied to hydraulics have developed rapidly in the recent past with the implementation of Volume of Fluid (VOF) and GIS techniques - even though there are still areas in which physical model tests are required for accurate results. The new technique of Smooth Particle Hydrodynamics (SPH) opens a new landscape for mathematical processing which gives good results in the more extreme situations of flow in hydraulic structures where many new possibilities can be explored.

SPH has shown good results in representing various coastal hydraulics phenomena but, will this type of mathematical processing also give good results in the more extreme situations of flow in hydraulic structures with high velocities. The hydraulic jump is a good test case because in this phenomenon turbulence plays a very important role and traditional models do not work.

In this paper a study is described in which the SPH approach is used to solve a case consisting of a high tank, to give energy to the water, regulated by a gate which is assumed to have an instantaneous opening. The floor downstream is horizontal and flat without roughness. A wide crested weir is used to force the jump. The case is similar to that of Forster and Skrinde, 1950.

For different jump shapes it was necessary to generate different upstream Froude numbers by adjusting the initial depth of water in the tank (H= 32, 18 and 10 m.), and the level of the weir (1, 2, 3, 4, 5 m.). The gate opening was in all cases fixed at one meter.

The validation of the results generated hitches - because the SPH test case is not stationary, in spite of the fact that the test results are in agreement with the theoretical calculation. This encouraged us to construct a physical model in a test flume to check the SPH result. Comparing the video record with the SPH animation a wonderful agreement was observed, as much in velocity of propagation of the wave, as in the level of the water downstream of the jump.

In conclusion, the SPH technique offers a powerful new tool for tackling hydraulics phenomena without physical model testing. At this time it is necessary to make an effort to develop this kind of model. In the near future this technological challenge will no longer be a problem.

KEYWORDS: SPH, HYDRAULIC JUMP

[1] Hydraulic Laboratory of Centro de Estudios Hidrográficos (CEH). CEDEX.
Pº Bajo de Virgen del Puerto nº3
28670- Madrid, SPAIN
David.lopez@cedex.es

1 INTRODUCTION

Mathematical models applied to hydraulics have developed rapidly in the recent past with the implementation of Volume of Fluid (VOF) and GIS techniques - even though there are still areas in which physical model tests are required for accurate results. The new technique of Smooth Particle Hydrodynamics (SPH) opens a new landscape for mathematical processing which gives good results in the more extreme situations of flow in hydraulic structures where many new possibilities can be explored.

SPH has shown good results in representing various coastal hydraulics phenomena but, will this type of mathematical processing also give good results in the more extreme situations of flow in hydraulic structures with high velocities. The hydraulic jump is a good test case because in this phenomenon turbulence plays a very important role and traditional models do not work.

2 THE SPH METHOD APPLIED TO FREE SURFACE FLOW.

The Smoothed Particles Hydrodynamics (SPH) method is a generic discretization technique for the continuum problem domain, an alternative from the difference, finite element or finite volume methods, with particles that move with the flow, transporting the flow characteristics in a Lagrangian way, without a grid. For a complete explanation of the method see Monaghan, 2005. From an exact integral representation, an arbitrary field function *A(r)*can be approximated in a discrete manner by particles of volume V_b and mass m_b and density ρ_b, as follows:

$$A(r_a) = \sum_b \frac{m_b}{\rho_b} A(r_b)\, W(r_a - r_b, h) \qquad (1)$$

Where r is the position vector and W a weighting function (called usually kernel), with characteristics like the Dirac´s delta. It has been chosen as a Gauss function where h is the parameter that represents the smoothed length, 20 cm in this study. From (1) is deduced, with some simplifications, an expression for the derivative:

$$A'(r_a) = -\sum_b \frac{m_b}{\rho_b} A(r_b)\, W'(r_a - r_b, h) \qquad (2)$$

It should be appreciated that the derivative is a function of the kernel. From (1) and (2) it is possible to present a discrete expression for the Navier- Stokes equations:

$$\frac{d\rho_a}{dt} = \sum_b m_b (v_a - v_b) \cdot \nabla W_{ab} \qquad (3)$$

$$\frac{dv_a}{dt} = -\sum_b m_b \left(\frac{p_b}{\rho_b^2} + \frac{p_a}{\rho_a^2} + \prod_{ab} \right) \nabla W_{ab}$$

equations for conservation of mass and momentum, respectively, where v is the velocity of the particles and p its pressure. The $\prod$ term represents a viscous acceleration useful for the treatment of shocks:

$$\Pi_{ab} = \begin{cases} \dfrac{-\alpha \bar{c}_{ab} \mu_{ab} + \beta \mu_{ab}^2}{\bar{\rho}_{ab}}, & v_{ab} \cdot r_{ab} < 0 \\ 0, & v_{ab} \cdot r_{ab} > 0 \end{cases} \qquad \mu_{ab} = \frac{h v_{ab} \cdot r_{ab}}{r_{ab}^2 + (0.1h)^2} \qquad (4)$$

α, β are parameters (usually α = 0.01 y β = 0, in free-surface flows), c is the sound velocity and $\bar{c}$ is an average value. The equations (3) have been chosen because their symmetric structure forces the algebraic performance of conservation. Thanks to the Lagrangian approach the convective terms disappear resulting in an ordinary derivative equation system. To close the problem, instead of considering the fluid uncompressible with a velocity field of divergence zero, Monaghan 1994, considers a fluid slightly compressive with a rigid equation which relates p and ρ.

$$P = \frac{\rho c_s^2}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right) \tag{5}$$

where C_s is the sound velocity and γ=7. So the compressibility is a function of the flow Mach number squared. Taking a value of the velocity c_s of about 10 times the maximum flow velocity gives a maximum change of density of about 1%. Integration over time is made by explicit method, such as Euler or predictor-corrector. Since the time step is controlled by the Courant number, it is interesting to take low values of the sound velocity (pressure wave propagation in flow).

Monaghan, 1992, presented a method to establish the boundary time step as a function of the Courant number, the viscosity and the particle acceleration. To calculate the interaction in each time step it is important to have a system for looking at the surrounding particles lineal with the number, a more exhaustive check method will be quadratic. An appropriate way to model quasi-uncompressible flow is to use a grid of proximity linked-list. Monaghan, 1992, proposes to use a correction XSPH, in the velocity of the particle movement, averaging with velocity of the surrounding particles. Therefore, two velocities coexist: one dynamic used in the equation of momentum, and another kinematic smoothed that is applied to the particles.

Some alternative ways exist to generate the boundaries in SPH. In this present work a method proposed by Monaghan, 1994 has been employed. This consists of simulating the boundary using particles with a repulsive force per unit of mass applied to the surrounding fluid particles.

$$f(r) = D\left(\left(\frac{r_0}{r}\right)^{p_1} - \left(\frac{r_0}{r}\right)^{p_2}\right)\frac{\vec{r}}{r^2} \tag{6}$$

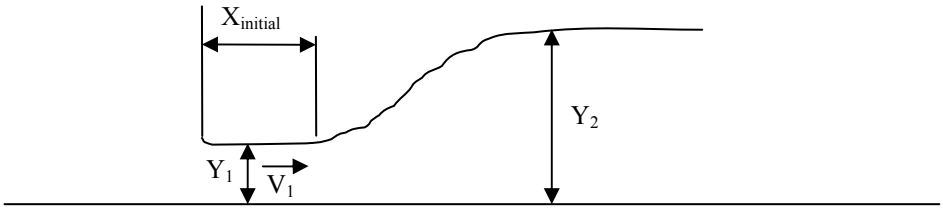
Where r_0 is the maximum distance reached by the force, $p_1 > p_2$ are coefficients (usually $p_1 = 4$ y $p_2 = 2$) and D a coefficient. This force represents a non-lineal elastic model. This means that a particle flowing parallel to the boundary is subject to a sinusoidal force. However, it doesn't distort significantly the results. We have used a distance between the boundary particles of 10 cm that means the half of the smoothed distance.

3 THE MATHEMATICAL TEST

In this paper a study is described where the SPH approach is used to solve a test case consisting of a high tank, to give energy to the water, regulated by a gate which is assumed to have an instantaneous opening. To make this study we have used an SPH FORTRAN code presented in Grassa (2004)

The floor downstream is assumed horizontal and flat with no roughness. A broad-crested weir is used to force the jump. The case is similar to that of Forster and Skrinde, 1950, in a study used to determine the minimum height of a weir needed to guarantee a hydraulic jump. To ensure that the water level downstream of the weir does not affect the jump, we have arranged a deep pool to collect the water.

For different jump shapes it was necessary to generate different upstream Froude numbers by adjusting the initial depth of water in the tank (H= 32, 18 and 10 m.), and the level of the weir (1, 2, 3, 4, 5 m.). The gate opening was in all cases set at one meter.



To limit the time of computation we have reduced the number of particles given special shapes to the tank, decreasing its volume and thus the number of particles. The stilling basin has been filled with particles. The particles in the tank were coloured red and those in the stilling basin coloured blue. The following figures show some of the tests at different times.

Test 1: $H_{Initial}$ = 10 m. Opening = 1 m. Weir depth = 1 m.

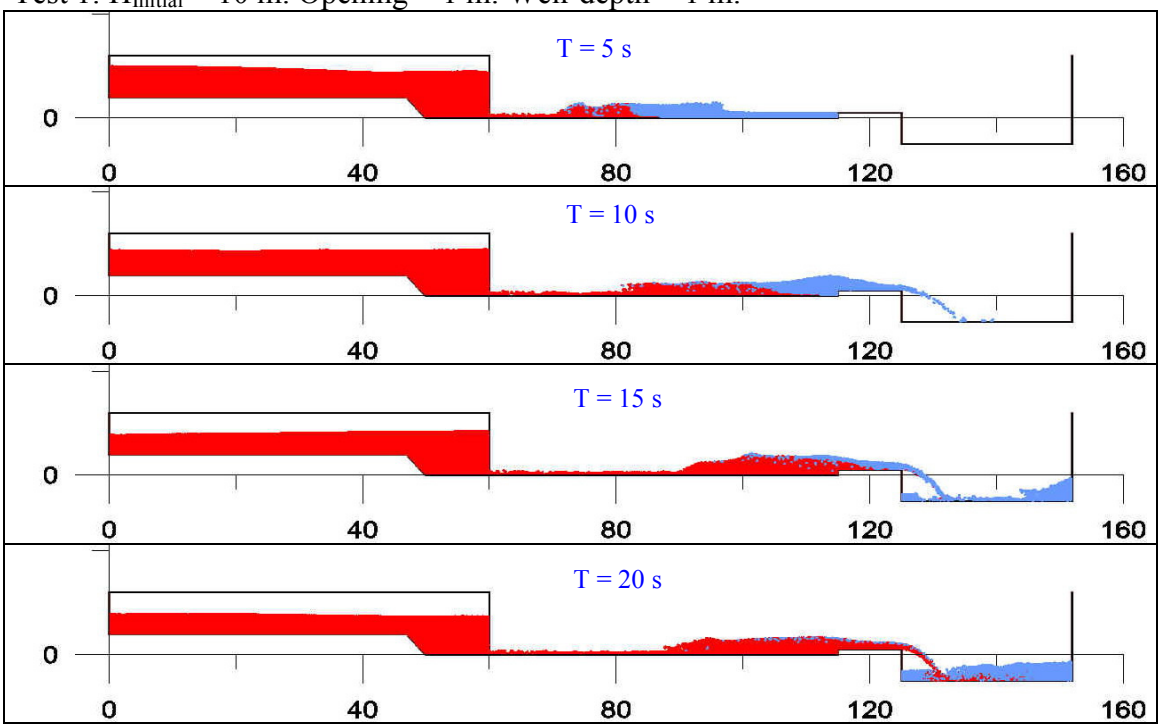


Figura 1

Table 1

Time (s)	Tank level (m)	Specific flow m3/s/m	$X_{initial}$ jump (m)	V_1 Speed (m/s)	y_1 Deep upstream (m)	F_1 Nº Froude upstream	y_2 Theoretical (m)	y_2 Deep downstream (m)
5	9.1	8.9	70	10.5	0.85	3.64	3.96	2.7
10	8.9	8.97	80	10.4	0.86	3.58	3.95	3.8
12	8.5	8.97	86	10.26	0.87	3.50	3.92	3.9
15	8.5	7.8	90	9.98	0.78	3.61	3.61	3.7
18	7.8	6.9	88	9.7	0.71	3.67	3.36	3.65
20	7.4	5.9	87	9.3	0.63	3.73	3.04	3.3

The test results are convincing. In the second t=5 after gate opening the flow pushes the water in the stilling basin giving rise to a mobile hydraulic jump traveling downstream since, the experimental downstream deep is bigger than the theoretical one, in accordance with:

$$y_2 = \frac{y_1}{2}\sqrt{1+8F_1^2} - 1 \tag{7}$$

At time, t= 10 the tail of the hydraulic jump arrives at the weir, generating the critical depth and imposing a higher level to the jump. Since this level doesn't arrive to the front of the hydraulic jump, it continues moving downstream. The level forced by the weir arrives to the front of the jump at t= 15. This one, together with the reduction of the flow due to the lower level in the tank, produces an experimental depth bigger than the theoretical one and the jump starts to move to upstream. This behaviour continues at time t= 20 and t=25.

Moreover, the shape of the jump in the test corresponds to the theoretical shape of a jump with $F_1 \approx 3.5$, an oscillating jump in agreement with the U.S. Bureau Reclamation classification.

In spite of the fact that the test results are in agreement with the theoretical calculation, the validation of the result generate hitches - because the SPH test is not stationary. This encouraged us to do a physical model to check the SPH result – which was built in a test flume.

Test 2: $H_{Initial}$ = 18 m. Opening = 1 m. Weir depth = 2 m.

Table 2

Time (s)	Tank level (m)	Specific flow m3/s/m	$X_{initial}$ jump (m)	V_1 Speed (m/s)	y_1 Deep upstream (m)	F_1 Nº Froude upstream	y_2 Theoretical (m)	y_2 Deep downstream (m)
5	16.5	9.95	68	14.9	0.67	5.82	5.18	4.5
10	16.4	10.7	76	14.63	0.73	5.46	5.30	5.5
15	15.7	9.78	78	14.18	0.69	5.45	4.99	5.3
20	14.25	8.88	69	13.46	0.66	5.29	4.63	4.7

Test 2: $H_{Initial}$ = 18 m. Opening = 1 m. Weir depth = 2 m.

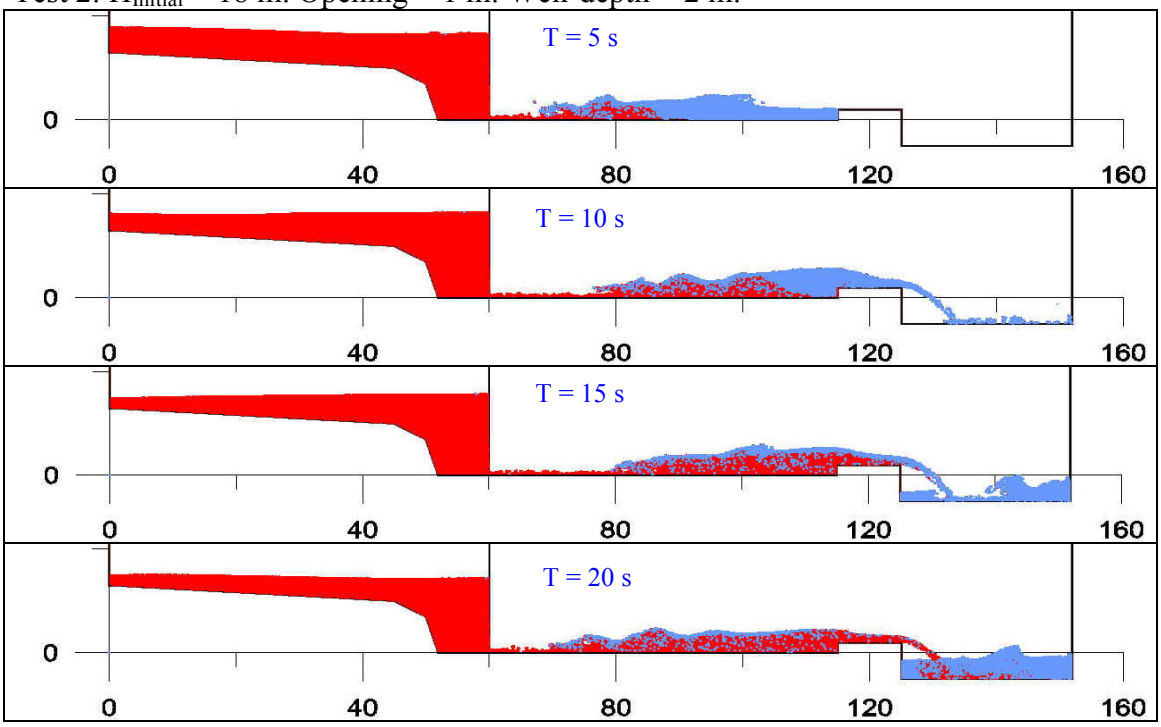


Figura 2

In this test the Froude number is around 5.5, which corresponds to the lower part of a permanent jump in agreement with the U.S. Bureau Reclamation classification.

Test 3: $H_{Initial}$ = 32 m. Opening = 1 m. Weir depth = 3 m.

Table 3

Time (s)	Tank level (m)	Specific flow m3/s/m	$X_{initial}$ jump (m)	V_1 Speed (m/s)	y_1 Deep upstream (m)	F_1 Nº Froude upstream	y_2 Theoretical (m)	y_2 Deep downstream (m)
5	30	12.9	66	20.39	0.63	8.19	7.02	4.98
10	29.7	13.48	73	20.29	0.66	7.959	7.14	7.9
15	28.76	12.17	73	19.65	0.62	7.98	6.68	8.3
20	26.7	11.5	70	14.4	0.801	5.15	5.43	5.7

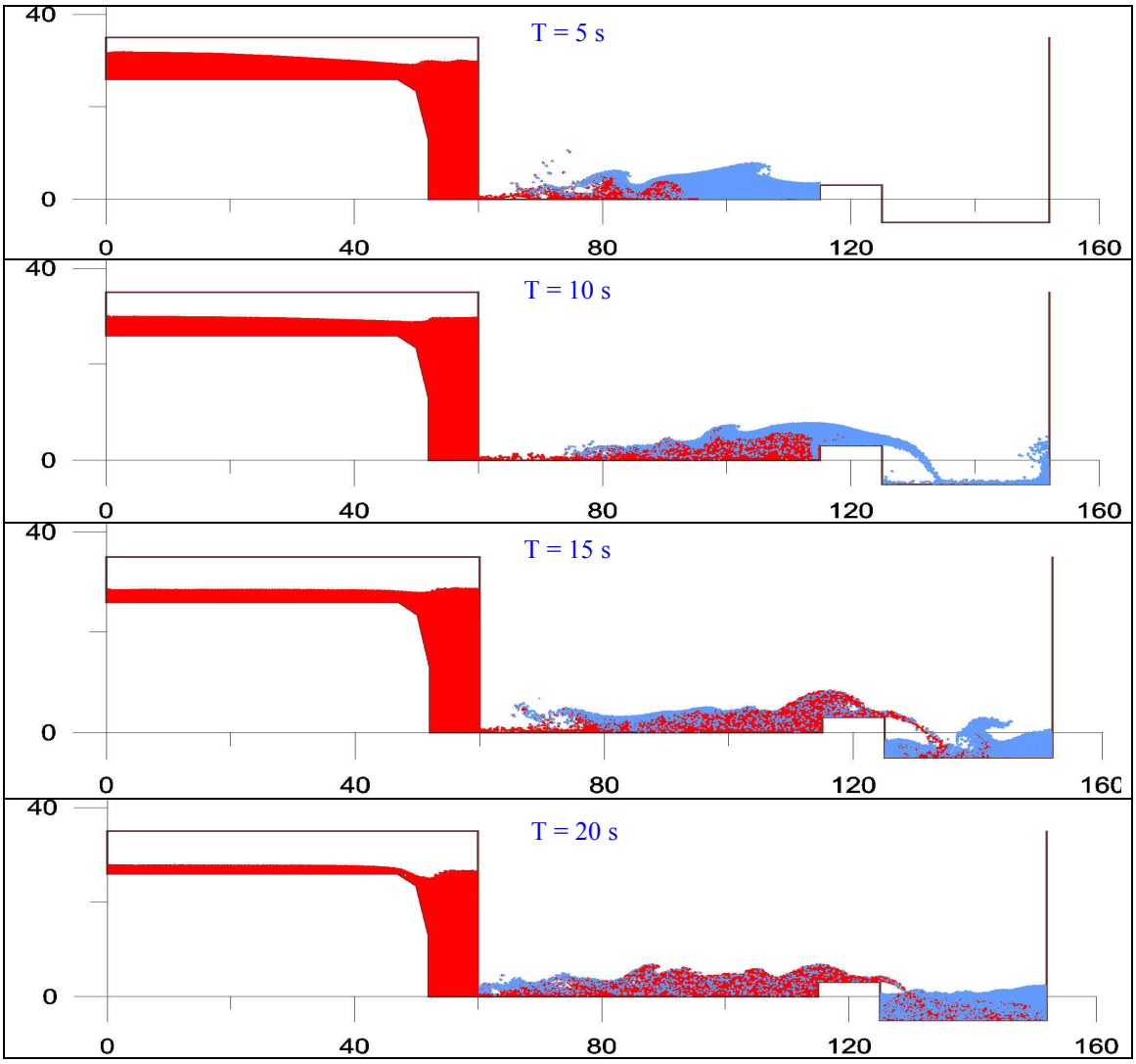
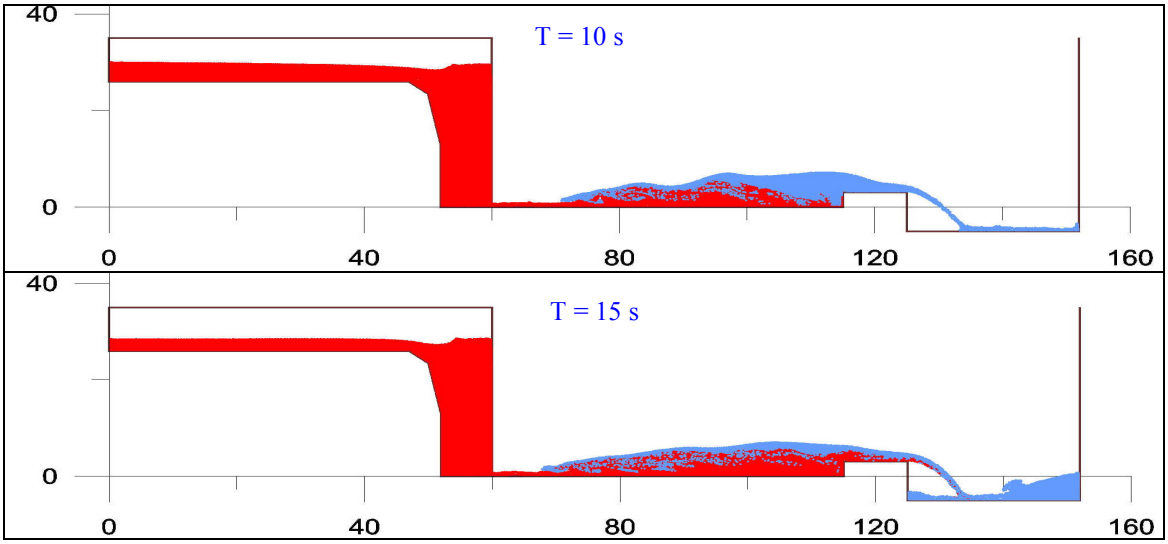


Figura 3

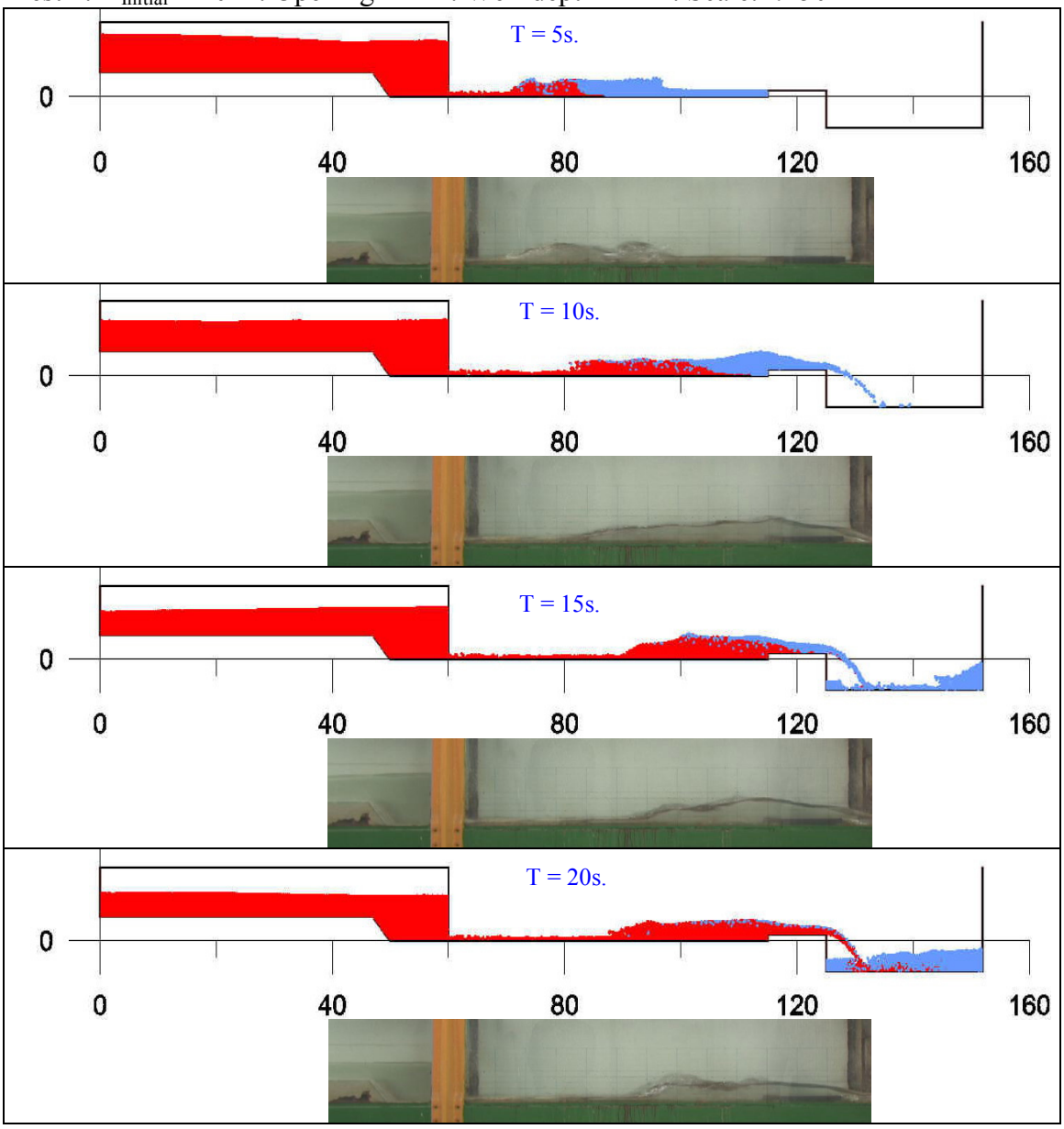
Like in the test 2 the free surface is more wavy than the a permanent jump (U.S.B.R. classification). We have eliminated that waves introducing in the calculation of $\prod$ the term $\beta\mu^2_{ab,}$ β=100



4 THE PHYSICAL MODEL TEST

The validation tests were carried out in a test flume of variable slope, even though the test was without slope. To adapt Test 1 to the flume geometry the scale used was λ=50. The time scale is $\sqrt{\lambda}$ for Froude similarity. Therefore, to compare the SPH animation with the movie record it is necessary to reduce the speed in this ratio. The following figure shows a good agreement between both tests.

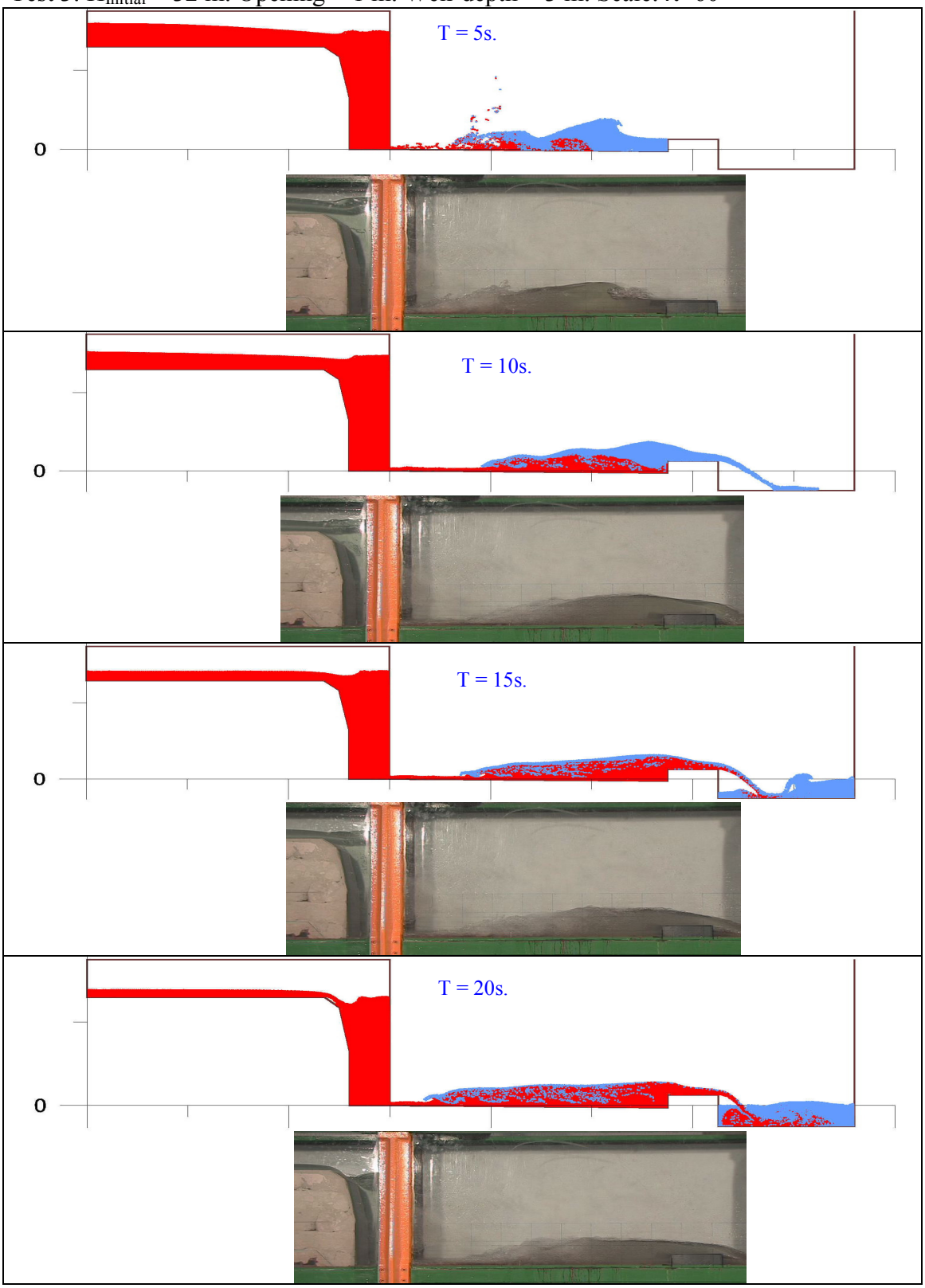
Test 1: $H_{Initial}$ = 10 m. Opening = 1 m. Weir depth = 1 m. Scale: λ=50



The frames comparison show a good agreement of the between the two models both in depth as well as in the speed of propagation of the jump.

The physical model to validate the test 3 was carry out with a floor slope of 1%, so it was necessary to repeat the SPH test to compare.

Test 3: $H_{Initial}$ = 32 m. Opening = 1 m. Weir depth = 3 m. Scale: λ=60



The Test shows a very good behaviour. The particles of differences colour let us appreciate the typical vortex of the jump.

5 CONCLUSIONS

In conclusion the SPH technique offers a powerful new tool for tackling hydraulics phenomena without physical model testing. We have shown that SPH has a very good agreement in jumps. To get a good agreement in flows with Froude number up to 5 it is better to take account of the β coefficient in the viscosity. It will be necessary to reproduce the roughness in SPH, and the permanent regime, besides the other challenges in SPH, such as 3D flow, as well as the computation capacity

6 ACKNOWLEDGEMENTS

I would like to thank , Jose Maria Grassa, Director of Coastal Hydraulics Center, CEDEX, Spain for his generous help.

7 REFERENCES

1. Colagrossi and Landrini, 2003. Numerical simulation of interfacial flows by smoothed particle hydrodynamics. Journal of Computational Physics, 191
2. Grassa, 2004. El método SPH. Aplicaciones en ingeniería marítima. Revista de Ingeniería Civil, 133
3. Grassa, 2006. Wave Forces on a Wavemaker. Sph Simulation and Comparison with Analytic Results. IAHR-LAD. Venezuela, 2006.
4. John W. Foster and Raymond A. Skrinde: Control of the hydraulic jump by sills, Transactions, ASCE, vol.115, pp.973-987, 1950.
5. Hugues, 1993. Physical models and laboratory techniques in coastal engineering. World Scientific.
6. Monaghan, 1992. Smoothed particle hydrodynamics. Annual review of astronomy and astrophysics, 30
7. Monaghan, 1994. Simulating free surface flows with SPH. Journal of Computational Physics, 110.
8. Monaghan, 2005. Smoothed Particle Hydrodynamics. Reports on Progress in Physics, 68.
9. Violeau and Issa, Numerical modelling of complex turbulence free-surface flows, with the SPH method: an overview. International Journal for Numerical Methods in Fluids 2007; 53:277-304.