Al-Azhar University Civil Engineering Research Magazine (CERM)
Vol. (43) No. (2) April . 2021

NUMERICAL STUDY FOR THE IMPACT OF SPILLWAY CONVERSION ON FLOW CHARACTERISTICS

Gamal M. Abdel Aal[(1)], Maha R. Fahmy[(1)], Eslam El-Tohamy[(1)]

and Marwa F. Shaheen[(2)]

[(1)]Water and Water Structures Engineering Department, Faculty of Engineering, Zagazig University
[(2)]Pyramids Higher Institute for Engineering and Technology

استخدام المحاكاه الرقمية لدراسة تأثير اختناق المفيضات على خصائص السريان

الملخص العربى:

المفيضات المختنقة قد تكون الحل الأفضل من الناحية العملية طبقا لموقع المفيض من السد. و لأن تشتيت الطاقة يعتبر الوظيفة الأساسية للمفيضات فتهدف هذه الدراسة الى تحديد تأثير الاختناق على خصائص السريان وبخاصة تشتيت الطاقة .وقد تم إستخدام نظرية التحليل البعدى لربط المتغيرات المختلفة التى تؤثر على خصائص السريان للمفيضات المختنقة ووضعها في صورة لا بعدية كما تم محاكاة التجربة ببرنامج flow 3d. هذا وقد توصلت الدراسة الى توافق جيد بين مخرجات البرنامج مع البيانات المعملية المقاسة. بالإضافة إلى ما سبق فإنه كلما زادت نسبة الاختناق للمفيض (نسبة عرض المجرى خلف المفيض إلى عرضه أمام المفيض) كلما قل الارتفاع النسبي للمياه في الخلف وكذلك طول القفزة الهيدروليكية بينما تزداد الطاقة المفقودة. وعند دراسة زوايا الاختناق وجد ان أفضل زاويه من حيث تشتيت الطاقه هى ($\theta\backslash 2=2.56^{O}$).

Abstract:

Conversed spillway may be some-times considered the best shape from the applicable point of view according to the dam site. The main function of spillway is energy dissipation. So studying the effect of the spillway conversion on the energy dissipation is very important, In this study the downstream width and the angle of conversion were tested experimentally.

The dimensional analysis was used to correlate the different parameters affecting the studied phenomena. Beside that the flow 3d model was used to simulate the data. . It was found that conversion decreases energy dissipation and increases downstream water depth. The numerical results were found to be in a good agreement with the experimental data. Finally, the results of this study could be recommended in the field of applications.

KEY WORDS: Energy dissipation, Experimental, Theoretical, conversion, Conversed spillway, entrance-

[(*)] Contact author (shaheen20007@yahoo.com)

1- Introduction

The large rise in water surface upstream the spillways generates a large potentional energy upstream spillways beside dams this energy causes scour in the spillway basin and the waterway downstream the dam , so many studies have been taken place to solve this problem and dissipate this energy to prevent scour and protect the waterway. When the water crosses the spillway the flow changes from subcritical flow upstream the spillway to super critical on and downstream the spillway. To return to sub- critical flow and form the tail water depth the hydraulic jump is formed, this hydraulic jump is the most effective way to dissipate energy. Many studies take place to maximize energy dissipation and control the length of the jump (A. J. Peterka 1958/1984) , (Mazumder 1988) , (Hager 1992) , (Christodoulou, G.C. 1993), (Chanson, H. 1994), (Yildiz 1998) ,(Chanson, H. 2001), (Jean G chatila 2004) ,(Hunt, S. L., and Kadavy, K. C. 2010a) & 2010b) and (FELDER, S., and CHANSON, H. 2011).

Upstream dams the water makes a large water storage area and this water crosses the dam through its spillway to the river basin which is smaller in wide than the storage area so we may need to use lateral constriction through or behind the spillway to reach the width of river basin as shown in Photo 1 this lateral constriction may be sudden or conversion (gradually contraction).

Kathleen Houston Frizell 1990 and his team constructed a physical model of Mc Clure spillway. This model aimed to identify the maximum capacity of existing spillway without over topping the side wall and propose another stepped spillway to contain the over flood. The proposed spillway was stepped spillway with step height from 1 to 4 feet and maximum conversion angle 12.68 degrees to satisfy the flow conditions and energy dissipation. Hunt 2008 studied a physical model to determine the maximum water height beside the convergence side walls for the design discharge. From the measured water surface the maximum water height equal three times the critical depth.(Hunt 2012) derive an equation for the water depth beside side walls and compared it with the experimental work.

Gamal M. Abdel Aal etal.,2018 [5] investigate the over-flow, , physical models of four steps were carried out on stepped spillways. Breakers were installed over the steps of spillway; So studying the effect of the breaker on stepped spillway on the energy dissipation Comparison of flow energy dissipation over the toe of stepped spillway with breakers and classical stepped spillway shows that the stepped spillway with breakers type had dissipated more energy than the classical stepped spillway type .**Reda M. Abd El-Hady Rady(2011)[10]**The study showed that the important variable governing discharge over sharp-crested weirs was Ht/tw. Further, the advantages of "Flow -3D" as a tool for examining velocity vectors and pressure pattern over rectangular sharp-crested weirs have been highlighted. **Reza Maghsoodi., etal. (2012)[11]**In this recent study we have simulated free surface flow over sub- merged weirs using three-dimensional (3D) flows were dominant. The fully 3D, Reynolds-averaged Navier– Stokes equations were solved using the model. the standard k–ε equations were used for turbulence model. The numerical results obtained using the compressed mesh systems were in accordance with the experimental data **Le Thi Thu Hien and Duong Hoai Duc (2020) [8**] they used Flow 3D with the Reynolds-averaged Navier–Stokes (RANS) and large eddy simulation (LES) turbulent models which included air entrainment to simulate the free surface flow We measured the water level, velocity and pressure to estimate the influence of grid size and the turbulent model type used. Our results highlight the need to include air entrainment in the model simulating rapid flow over a hydraulic construction. With adjustments for energy loss, this study shows that walls provide the best results and the optimal distance between two walls is 2.8 m.**Babak Ghazi , etal., (2019) [4]** the three-dimensional model of Shahid Madani Dam's spillway was simulated with the Flow 3D software and by the comparison of numerical model results with the experimental data, **Mehdi Taghavi, Hesam Ghodousi(2015) [**9] Using different geometrical and hydraulic conditions, the discharge coefficient relation in sharp-crested and side channel weirs for the case of suspended load can be extracted. Flow3D numerical model is capable of precisely simulation the flow containing suspended and bed load in the main channel.

Hamed Sarkardeh etal., [6] In the present study, a physical model as well as a numerical model was employed on a case study of stepped spillway to modify the transitional zone and improve flow pattern over the spillway. RNG model and for free surface VOF model was selected in the numerical model. Results of the numerical and physical models were compared and good agreement concluded in flow conditions and energy dissipation .**REZAZADEH Shiva etal., (2020)** [12]The purpose of this research is simulate the flow over sharp crested weir and investigate the effect of geometric shapes of sharp crested weirs on hydraulic characteristics of the flow such as pressure, velocity, water level profiles and discharge coefficients. Thus the limitation and usage range of sharp crested weirs are clarified. In this research Open FOAM open source 3D software with RNG K-ε turbulence model and Volume of Fluid method (VOF) was used to analyze the hydraulic flow passing through sharp crested weir. **Shaymaa A. M. Al-Hashimi etal.,(2017)[14]** In this study, FLUENT software as a type of Computational Fluid Dynamics (CFD) model, represented .The volume of fluid (VOF) method with four turbulence models of Standard k–ε, RNG k–ε, Realizable k–ε and Standard k–ω are applied to estimate the free surface profile. **Saleh I. Khassaf , etal.,(2016)[13]** In this study, the coefficient of discharge over the two compound side weirs (Rectangular and Semi-Circle) were modeled by using Computational Fluid Dynamic (CFD) to describe the flow characteristics in subcritical flow conditions. (Flow-3D) program was used to determine the numerical uncertainty of the simulation results.

Hossein Afshar, Seyed Hooman Hosei(2013)[7]In this study, Computational Fluid Dynamics (CFD) model together with laboratory model were used in order to determining the free-surface profile of rectangular broad-crested weir. Simulations were performed using the volume of fluid (VOF) free surface model and three turbulence models of the

RNG k–ε, standard k–ε and the large eddy simulation (LES) to find the water level profile and streamlines. The computational results showed a close agreement with experimental data obtained in the laboratory. Also, results indicate that RNG model has the minimum level.**A**.**Ghafourian,etal.,(2011)[1]**At present the hydraulic of siphon spillway usually is done by physical modeling.. At the same time an available CFD program, which solves the Navier-Stokes equations, was run by using data obtained from physical model. The result of investigations shows that: Comparison of discharge coefficient in pressurized spillway with both free and submerged outlet shows that submerging in outlet is effective in promotion of spillway efficiency. **Abdul-Hassan K. Al-Shukur etal., (2014)** The results showed that the using of different types of steps have the most influence on the amount of energy
losses, they are increased the energy losses by about (1.6%-18.6%) at different runs and configurations in comparison of horizontal steps. The results indicated also that the energy dissipation rate at configuration of alternative inclined and horizontal steps show increased by about (6.2% at q=0.02119 m3/s/m) more than the configuration of alternative endsills and horizontal steps and by about (5.3% at q=0.02119 m3/s/m) more than configurations of inclined on all steps. **Amir Reza Razavi , Hassan Ahmadi , (2017)** numerical model of FLOW-3D can provide valuable information in this regard. In the present study, flow was calibrated and validated using FLOW-3D through physical model The research findings revealed that increasing suspended flow load leads to decreasing values of flow passing through the morning glory spillways; such that, deceased values strongly depend on suspended load.

In this research the effect of the conversion angle and contraction ratio through the spillway on energy dissipation, tail water depth and jump length were studied.

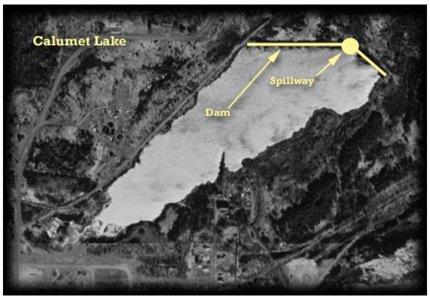


photo (1) Calumet Lake Ref.[14]

2-Experimental setup

Experimental work was carried out in the Hydraulic and water Engineering Laboratory of the Faculty

of Engineering, Zagazig University, Egypt. The used flume is 30 cm width, 46.8 cm depth with an overall length of about 15.6 m. The flume is equipped with a tailgate to control the tail water depth. A pump lifts the water from a tank to the flume inlet. The water runs through the flume working section then returns back to the tank in a closed circle. The discharge was measured by a pre-calibrated orifice meter installed in the feeding pipe line.

The used spillway model is made from wood with 24.5 cm height and 30^{O} angle slope with the horizontal bed as shown in Figure 1. Different contraction ratios (C= b/B, i.e.100 %, 83.33%, 76.67%, and 70%) through the spillway were tested under the same flow conditions and the same angle of conversion $\theta/2 = 1.65^{O}$. Then the effect of conversion angle with $\theta/2 = (1.65^{O}, 2.00^{O}, 2.65^{O}, 6.50^{O}$ and $90^{O})$ were tested for the contraction ratio C=70% and the same flow conditions.The conversion side walls were made of poly-carbonate sheets supports by wood bars were used as shown in photo 2.

Photo (2) Experimental model

3- Dimensional analysis

Using the principals of the dimensional analysis, the different variables affecting the energy dissipation through the spillway could be formulated in the following dimensionless equation:

$$\frac{\Delta E}{E_{up}} = f(F_{up}, \frac{Y_2}{Y_{up}}, \frac{L_j}{Y_2}, C, \theta)$$

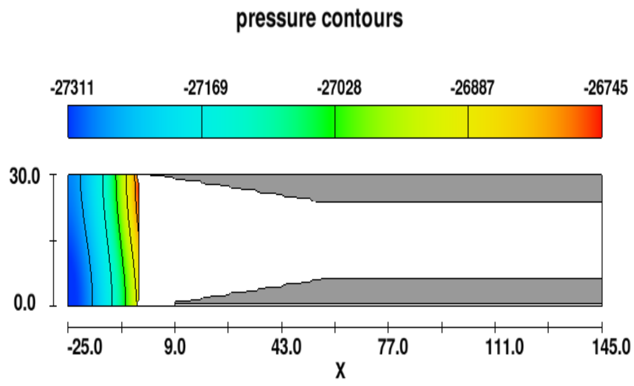
Where

C is the contraction ratio, b is the contracted width, B is the width of channel, θ is the angle of divergence, Y_{up} is the upstream water depth, Y_2 is the downstream water depth, L_j is the jump length and Fup is the upstream Froude number

4- Numerical model.

Using flow 3d model experimental results were simulated to test the applity to apply such models in the program.

The resultant data were accurate with accepted error ± 10



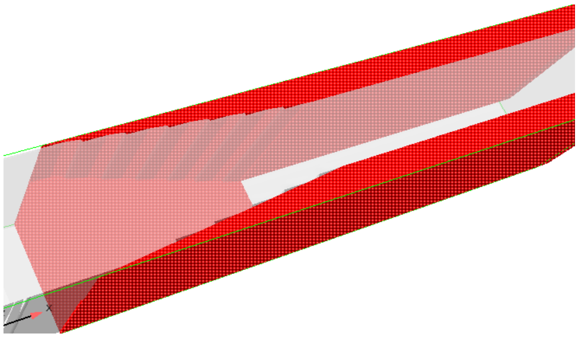


Photo (3) flow 3d model

5- Analysis of Experimental Results

The different flow characteristics were analyzed as a function of the inflow Froude number .Such as the energy loss, jump length, tail water depth and the bed width contraction.

5.1. Effect of contraction ratios.

Fig.1 represents the relationship between the upstream Froude number F_{UP} and the relative energy loss $\Delta E/Eup$ for different contraction ratios. From this figure, for the same contraction ratio, as Froude number increases the relative energy loss decreases. Moreover, for the same Froude number, the relative energy loss decreases as contraction ratio decreases. The relative tail water depth plotted against the U.S. Froude no. as shown in Fig. 2. From this figure, the relative tail water depth increases as the contraction ratio decreases.

In non-prismatic stilling basin the jet of flow spread in the vertical and lateral direction. In conversed stilling basin the lateral direction is gradually contracted, then the flow jet spreads in vertical direction more than non-prismatic case and gains potential energy. This causes increasing the tail water depth and decreasing the energy loss. In addition as the contraction ratio decreases, the flow jet takes place more vertical transition, higher tail water depth and less energy dissipation. Moreover, for the same flow condition as the upstream water depth increases, the flow velocity increases and also the jump length. On the other side as the upstream water depth increases Froude number decrease

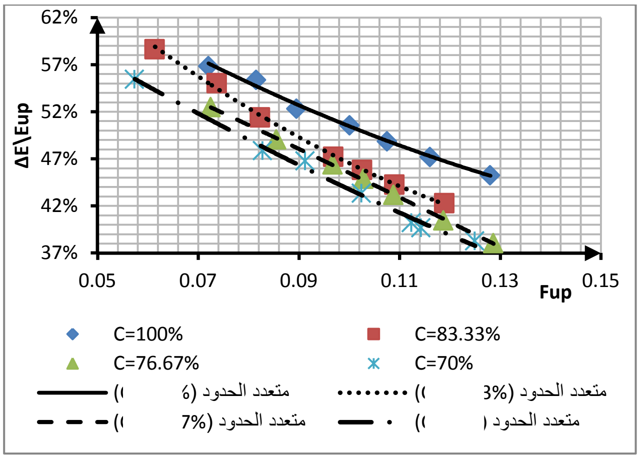


Figure (1). Relations between F_{UP} and $\Delta E/E_{up}$ for different contraction ratios.

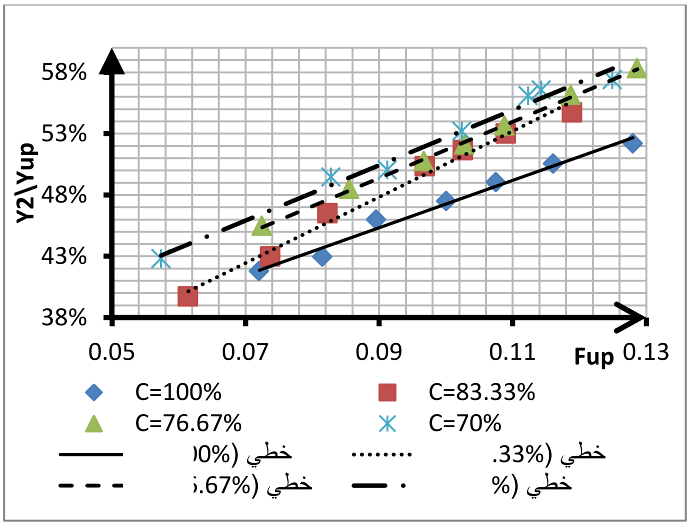


Figure (2). Relation between F_{UP} and Y_2/Y_{UP} for different contraction ratios

5.2. Effect of the conversion angle

As shown in (Fig.4) the case of (θ\2=2.00°) and (θ\2=2.56°) give the smallest relative flow depth Y_2/Y_{UP}, otherwise in (Fig.3) the two angles gives the max energy dissipation. Energy dissipation $\Delta E/Eup$ decreases for angles less than (θ\2=2.00°) or larger than (θ\2=2.65°) and the minimum energy dissipation occurs for (θ\2=90°).

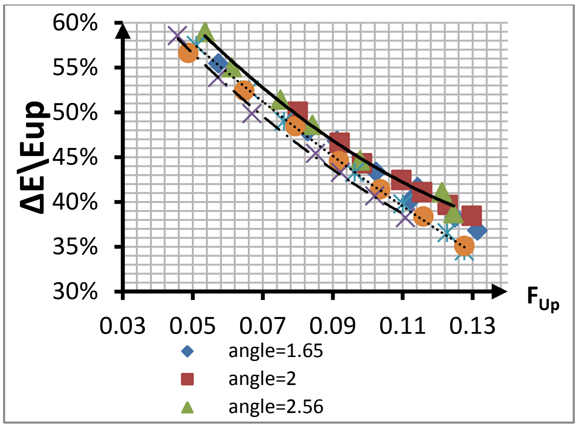


Figure (3) Relations between F_{UP} and $\Delta E/E_{up}$ for different conversion angles.

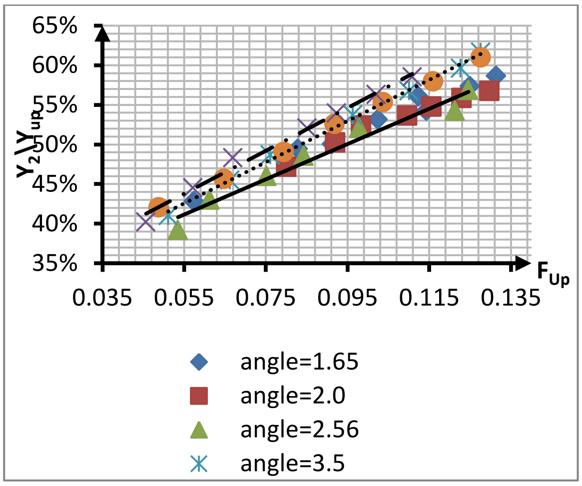


Figure (4) Relation between F_{UP} and Y_2/Y_{UP} for different conversion angles.

6- Verification.

The experimental results were used to verify the numerical results

From (Fig.5), a comparison could be shown between the predicted and measured Froude number with maximum error ±10%.

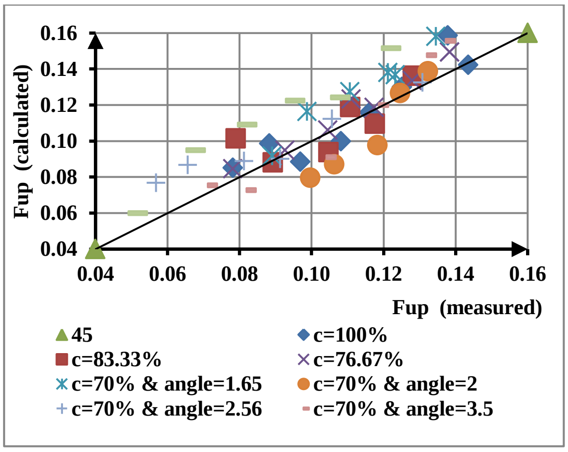


Figure (5). Comparison between experimental results and numerical model results

7-Conclusions

The present study introduced the following results.

1- The conversion decreased energy dissipation and increased downstream water depth and jump length.

2- As the contraction ratio increased, the tailwater depth and the jump length decreased and the relative energy loss increased.

3- The conversion angle of ($\theta\backslash 2=2.00^{O}$) and ($\theta\backslash 2=2.56^{O}$) produced the smallest relative flow depth Y_2/Y_{UP} and jump length, and the largest relative energy loss $\Delta E/Eup$

4- The numerical results of energy dissipation were in a good agreement with relative error (± 10%) with the experimental results, so the derived equations can be recommended in the field of applications.

8- References

[1] A. Ghafourian, H. Mousavi jahromi and M. Shafaee Bajestan, (2011), "Hydraulic of Siphon Spillway by Physical and Computational Fluid Dynamics", World Applied Sciences Journal 14 (8): 1240-1245, 2011 ISSN 1818-4952

[2]Abdul-Hassan K. Al-Shukur ,Safaa K. Hashim Al-Khalaf , Ishraq M. Ahmed Al-sharifi (2014)" Flow Characteristics and Energy Dissipation Losses in Different Configurations of Steps of Stepped Spillway " International Journal of Innovative Research in Science,Engineering and Technology(An ISO 3297: 2007 Certified Organization)Vol. 3, Issue 1, January 2014

[3] Amir Reza Razavi , Hassan Ahmadi (2017). " Numerical Modelling of Flow in Morning Glory Spillways Using FLOW-3D" Civil Engineering Journal Vol. 3, No. 10, October, 2017

[4]**Babak Ghazi, Rasoul Daneshfaraz, Esmaeil Jeihouni (2019)** " Numerical investigation of hydraulic characteristics and prediction of cavitation number in Shahid Madani Dam's Spillway " Vol.7 No.4: 323—332,Iran

[5] Gamal M. Abdel Aal, M. Sobeah, Esam Helal, M. El-Fooly (2018). "Improving energy dissipation on stepped spillways using breakers" / Ain Shams Engineering Journal 9 (2018) 1887–1896

[6] Hamed Sarkardeh, Morteza Marosi, Raza Roshan(2015) "Stepped spillway optimization through numerical and physical modeling " Volume 6, Issue 6, 2015 pp.597-606, Iran

[7] Hossein Afshar, Seyed Hooman Hosei (2013). "Experimental and 3-D Numerical Simulation of Flow over a Rectangular Broad- Crested Weir."

International Journal of Engineering and Advanced Technology (IJEAT)ISSN: 2249 – 8958, Volume-2, Issue-6, August 2013

[8] Le Thi Thu Hien 1,* and Duong Hoai Duc(2020) "Numerical Simulation of Free Surface Flow on Spillways and Channel Chutes with Wall and Step Abutments by Coupling Turbulence and Air Entrainment Models "Journal of Groundwater Science and Engineering,Vol.7 No.4: 323—332 Vietnam

[9] Mehdi Taghavia, Hesam Ghodousib (2015)," Simulation of Flow Suspended Load in Weirs by Using Flow3D Model " Vol. 1, No. 1,November,,Iran
Canal Falls"J.of C.E. Div.,The Institution of Engineers (India) Vol. 68 Pt. CI 6 ,May,1988

[10] Reda M. Abd El-Hady Rady (2011). "2D-3D Modeling of Flow Over Sharp-Crested Weirs." Journal of Applied Sciences Research, 7(12): 2495-2505, 2011 ISSN 1819-544X

[11] Reza Maghsoodi, Mohammad S. Roozgar, Sarkardeh, and Hazi M. Azamathulla(2012)" 3D-SIMULATION OF FLOW OVER SUBMERGED WEIRS" International Journal of Modelling and Simulation, Vol. 32, No. 4, 2012

[12]Rezazadeh shiva manafpour mohammad , Ebrahimnejadian hamze(2020), "THREE-DIMENSIONAL SIMULATION OF FLOW OVER SHARP- CRESTED WEIRS USING VOLUME OF FLUID METHOD "International Energy and Environment Foundation,2015

[13] Saleh I. Khassaf1, Ali N. Attiyah, Hayder A. Al-Yousify(2016)" Experimental investigation of compound side weir with modeling using computational fluid dynamic"Iraq

[14] Shaymaa A. M. Al-Hashimi1, Huda M. Madhloom, Thameen Nazar Nahi(2017)" Experimental and Numerical Simulation of flow over broad crested weir and stepped weir using different turbulence models"Vol. 21, No. 02, March 2017
ISSN 2520-0917www.jeasd.org