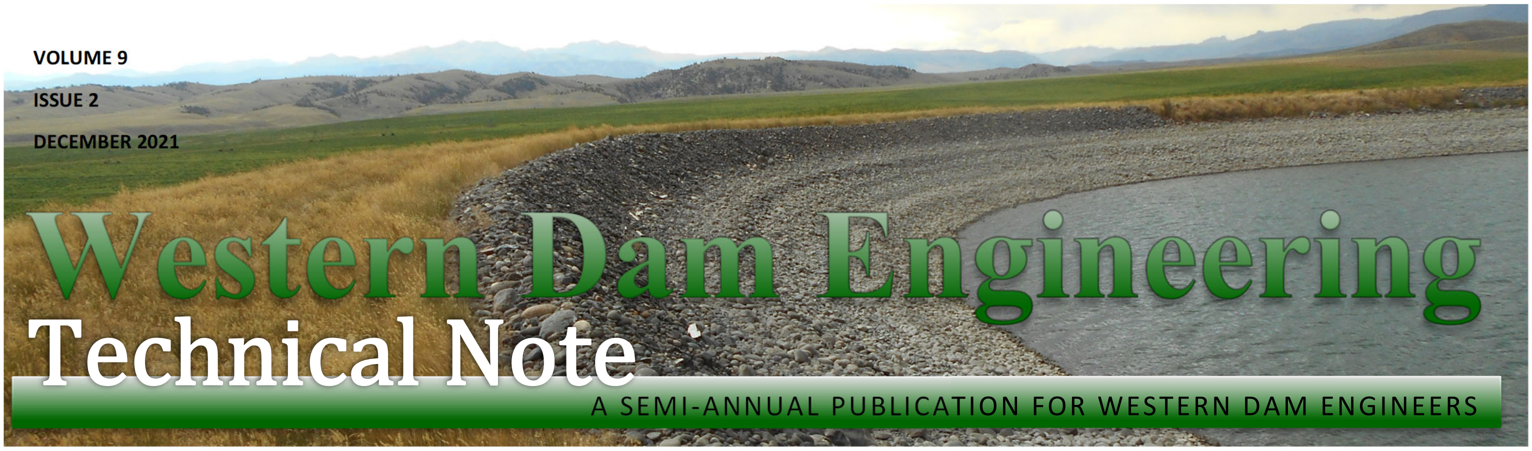


VOLUME 9

ISSUE 2

DECEMBER 2021

Western Dam Engineering Technical Note

A SEMI-ANNUAL PUBLICATION FOR WESTERN DAM ENGINEERS

In this issue of the ***Western Dam Engineering Technical Note***, we present articles introducing **seepage modeling** and **computational fluid dynamic modeling**. This newsletter is meant as an educational resource for civil engineers who practice primarily in rural areas of the western United States. This publication provides general technical information focusing on small and medium dams. The reader is encouraged to use the references cited and engage other technical experts as appropriate.

Good to Know

Upcoming ASDSO Webinar Dam Safety Training:

- *The History of Dam Safety Governance in the US Including Risk Analysis, January 11, 2022*
- *Specialty Construction Techniques for Foundation Improvement and Seepage Reduction, February 8, 2022*
- *Findings of the Independent Forensic Investigation of the Failures of Edenville and Sanford Dams, Michigan, March 8, 2022*
- *Concrete Construction – Issues and Remedies, April 12, 2022*
- *Inspection and Assessment of Spillways, May 10, 2022*

Upcoming ASDSO Virtual Technical Seminars:

- *Improving Emergency Operations for Dam and Levee Failures and Incidents, January 25-28, 2022*
- *Dam Construction Inspection, February 9-11, 2022*
- *Inspection & Assessment of Dams, March 22-24, 2022*
- *Seepage Through Earth Dams, April 26-29, 2022*
- *Earthquake Engineering for Embankment Dams, May 23-25, 2022*

Conferences:

- *West Regional Conference, Layton, UT, February 15-18, 2022*

ASDSO Training Website Link

The Western Dam Engineering Technical Note is sponsored by the following agencies:

- Colorado Division of Water Resources
- Montana Department of Natural Resources
- Wyoming State Engineer's Office
- New Mexico Office of the State Engineer

This Technical Note was compiled, written, and edited by AECOM in Denver, CO.

Funding for the Technical Note has been provided by the FEMA National Dam Safety Act Assistance to States grant program.

In this Issue

What the Flow!! Analyzing Seepage in Embankment Dams

By: Harry C. Donaghy, PE

Introduction

Seepage in embankment dams occurs in many forms and can sometimes be cause for frustration and even alarm. You may find yourself at the base of a dam asking…what the flow? If this does happen it's important to take a closer look at the seepage and its effect on dam safety. Seepage analysis allows engineers to evaluate the potential for destabilizing or erosive forces in a dam. This article discusses the methods used in current practice and provides tips for planning and interpreting seepage analyses for embankment dams.

Seepage is the flow of water through the porous space within a soil or rock mass. In embankment dams, seepage can occur through the embankment, foundation, abutments, or along embankment penetrations. This includes flow through a large area of soil or concentrated flow along defects, such as cracks, loose lifts, rock discontinuities (e.g., fractures and joints), and other pathways. The reservoir generally provides the largest source of water for seepage, but it may also come from groundwater sources. Figure 1 shows an example of seepage emanating from the downstream toe of an embankment dam. Almost all dams seep or leak, so the trick is to collect, control, and convey seepage safely away from the dam. Measurement of seepage and change in flows is critical, as is monitoring for sediment, which can include suspended and dissolved solids.

Seepage and leakage are not necessarily a problem if it is identified, monitored, evaluated, and controlled. However, seepage can become a serious dam safety concern if it is not controlled, especially since it is an initiating factor for potential failure modes related to internal erosion, excess uplift pressures, or instability. Can you tell if the seepage, shown in Figure 1 is a dam safety concern, or simply a maintenance issue? If not, a seepage analysis may help inform you.

Figure 1: Seepage Emanating from Downstream Toe of Embankment Dam (Photo Courtesy of the Department of Natural Resources and Conservation)

What Is a Seepage Analysis?

A seepage analysis is a computational method that models the seepage conditions of an embankment dam to estimate seepage characteristics through, beneath, and/or around the embankment. A seepage analysis can provide an understanding of the following:

- Seepage pathways,
- Seepage flow rates and velocities,
- Seepage gradients,
- Total head, pressure head, and pore water pressures, and
- Saturation.

Common applications of a seepage analysis include quantifying the performance of the dam under current, expected future, and/or extreme conditions; evaluating observed seepage conditions; or evaluating and comparing seepage control design alternatives.

Seepage Analysis Methods

There are a variety of seepage analysis methods for evaluating embankment dams that range from simple graphical approaches, completed by hand, to more complex numerical modeling using computer programs. Selecting an appropriate analysis method will depend on the objective of the seepage analysis and complexity of the situation. Most methods incorporate Darcy's Law and involve solving the Laplace equation. Seepage analysis methods include the following:

- Graphical Methods
  - Flow Nets
  - Graphical Construction of Phreatic Surface
- Numerical Models

Generally, it is best to start with the simplest and least expensive method before proceeding to a more complex and costly method. However, powerful, user-friendly, and commercially available seepage software programs have made the use of numerical modeling the current standard of practice.

Graphical Methods

Analyzing the seepage response of embankment dams using 2D graphical methods is the oldest approach and includes creating flow nets or graphically constructing the phreatic surface.

Flow Nets

A flow net is constructed by hand and consists of two sets of orthogonal curves referred to as flow lines and equipotential lines. Flow lines represent seepage paths through the soil and trend from the reservoir to the downstream toe. The space between two flow lines is a flow channel, and each channel represents equal quantities of flow. Equipotential lines intersect the flow lines at right angles. Equipotential lines show the location of points within the soil that have the same piezometric head and represent equal pressure drops along the flow net. Figure 2 illustrates a flow net for unconfined seepage through a homogeneous embankment with a rock fill toe drain.

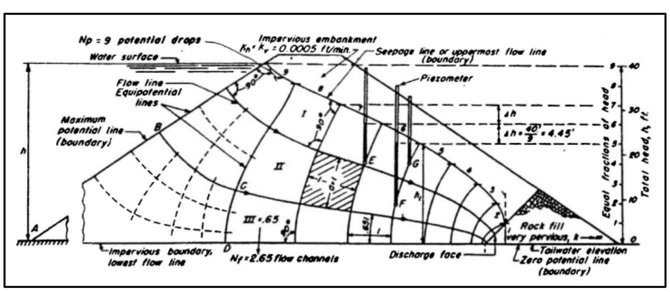


Figure 2: Flow Net for Unconfined Seepage through a Homogeneous Embankment (Adapted from [1])

Flow nets are a practical and versatile method for evaluating seepage and have historically been used to analyze 2D seepage problems. Flow nets are relatively fast, easy to draw for simple cases, inexpensive, and provide insight into seepage flow characteristics and quantities. However, flow nets take practice and experience to draw accurately, require a fair amount of simplification to geometry and material properties, and are difficult to draw for complicated geometries and multiple permeabilities. Although they have become less widely used, with practice, flow nets can be a valuable tool for understanding seepage in dams and can also be used to help verify numerical solutions. For more information on flow nets, refer to [1], [2], and [3].

Conceptual Flow Net

Modelers should attempt to draw a conceptual flow net in advance of modeling, as it will help guide thinking and aid in model set up and decision making. A conceptual model can be developed in AutoCAD, Excel, or any software that maintains geometric accuracy. Hand-drawn models on graph paper are also acceptable. Plot the cross section of the dam, materials, piezometers (and zones of influence), maximum reservoir level, and associated measured water levels in the piezometers. Using the known values of total head and your knowledge of the dam, draw approximate flow paths and total head contours. **If you lack sufficient information to draw a conceptual model, you likely do not have enough information to justify a numerical modeling effort.**

Graphical Construction of Phreatic Surface

The upper line of seepage (i.e., flow line) through an embankment dam is known as the phreatic surface and represents a line of zero pressure. The phreatic surface through an embankment can be graphically constructed. Figure 3 illustrates the graphical construction of a phreatic surface for a homogeneous embankment.

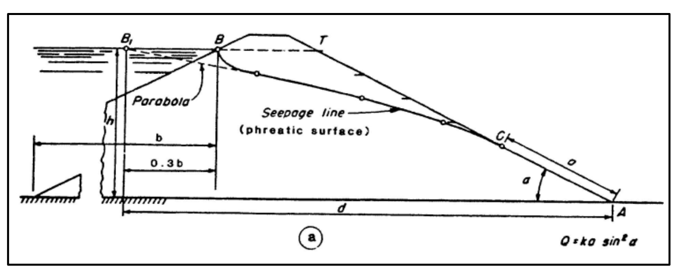


Figure 3: Graphically Constructed Phreatic Surface through a Homogeneous Embankment with Drainage Blanket (Adapted from [1])

Graphically constructing the phreatic surface is a relatively fast, easy-to-follow, and inexpensive approach; it uses historical piezometer levels to define the phreatic surface for slope stability analyses or estimating seepage quantities. This approach is fast, repeatable, and may also be used as a starting point for construction of a flow net. However, it has limited applicability, less versatility than flow nets, and is no longer commonly used. Graphical construction of the phreatic surface is primarily limited to evaluating drainage alternatives for homogeneous embankments on relatively impervious foundations. For more information on graphically constructing the phreatic surface, refer to [1], [2], and [3].

Numerical Models

Analyzing the seepage response of embankment dams using 2D and 3D numerical models is the most common approach and current standard of practice.

Numerical models are a finite element analysis that use computer programs to mathematically approximate the Laplace equation in complex flow conditions. In a numerical model, the geometry is discretized into small (i.e., finite) elements that form a grid. Each element intersection is called a node, and the nodes represent a continuum through the entire model. The model uses a series of equations to approximate the Laplace equation. For example, if the grid consists of N elements, there will be N equations and N unknowns to solve. Figure 4 illustrates a numerical seepage model for a zoned embankment.

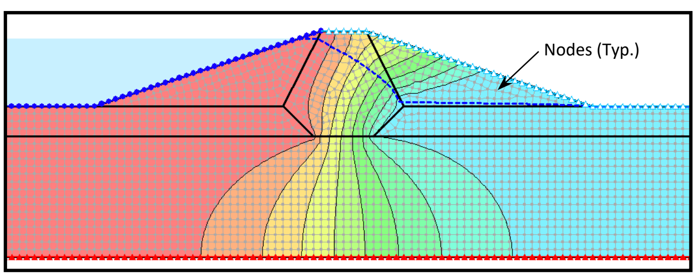


Figure 4: Numerical Seepage Model for Zoned Embankment

There are many benefits to numerical modeling, which include the following:

- Numerical models can characterize permeability and evaluate flow through both saturated and unsaturated soils. Characterizing unsaturated flow is a limitation of flow nets.
- Numerical models are easier to use for complex situations and zoned dams (e.g., complex embankment geometry or foundation stratigraphy).
- Both steady-state and transient (or time-dependent) flow can be modeled.
- Both 2D and 3D problems can be modeled.
- Zones where seepage gradients or velocities are high can be more accurately modeled by varying the size of the discrete elements.
- A variety of boundary conditions can be modeled, which may be useful for prediction of seepage flow under a future loading condition (i.e., a higher reservoir level than has been experienced before).
- Most numerical models have graphical results that can be visually checked for reasonableness and better communicate the results to lay people.
- Numerical models provide results (seepage flow rates, velocities, gradients, pressures, etc.) at any location (i.e., node) within the model.
- Results can be easily used and input into slope stability computer programs.

Some limitations to numerical modeling include the following:

- Numerical models are only as good as the modeler's veracity, understanding of the input, and ability to interpret the results.
- Modeling requires practice and training to understand the sensitivities of the model.
- Numerical models are susceptible to convergence issues.
- Numerical models will often run without error and produce professional-looking results that can be invalid or produce results that do not make sense. It takes knowledge of seepage principles and experience with modeling to properly interpret and verify the results.
- Modeling can be time-consuming and costly.

For more information on numerical models, refer to [4] and [5].

The remainder of this article focuses on the planning and interpretation of numerical modeling because it is the current standard of practice in the industry.

When Is a Seepage Analysis Warranted?

A seepage analysis of an embankment dam can be used to gain a holistic understanding of the seepage regime through the embankment and its foundation. Seepage modeling can also provide valuable insight into how seepage may influence the performance of the embankment dam. However, seepage modeling can be very time-consuming and costly and may not provide any additional insight into seepage-related issues if it is not warranted. Key considerations in evaluating the cost-effectiveness and necessity for a numeric seepage model include the objective of the analysis, the amount of available information, and the complexity of the seepage regime. To identify when a seepage analysis may be warranted, consider the following questions:

- What is the objective of the seepage analysis?
- Can the objective be met adequately without a seepage analysis?
- Will a seepage analysis accurately capture the objective?
- Is there time to complete a seepage analysis (not an emergency)?
- Is there enough information available on the characterization of the materials to perform an effective analysis?

What Is the Objective?

When establishing the objective of a seepage analysis, preferably before performing the analysis, it is important to have a basic understanding of when a numerical seepage model may or may not be warranted for an embankment dam. A few examples that may or may not warrant a numeric seepage model are summarized in Table 1.

Table 1: Examples That May or May Not Warrant a Numerical Seepage Model

Examples that May Warrant a Numeric Seepage Model	Examples that May Not Warrant a Numeric Seepage Model
Evaluating an embankment under a future reservoir operating or flood loading condition it has not yet experienced.	Site with limited information on the embankment zoning, foundation conditions (including geology and stratigraphy), and material properties.
Evaluating an embankment with a complex geometry or foundation stratigraphy and/or a site with complex geology.	Well-instrumented site (e.g., piezometers) in which current data can be used to evaluate seepage response rather than a model.
Evaluating a non-hydrostatic condition (i.e., total head is not constant with depth).	A site that is not sensitive to seepage performance, which can be demonstrated using conservative assumptions.
Site configuration that limits accuracy of 2D simplified assumptions.	Sensitivity analyses (e.g., slope stability) can be performed for a potential range of pore water pressures to compensate for uncertainty.

Can the Objective Be Met without a Seepage Analysis?

In some cases, the specified objective can be met without a seepage analysis if the embankment dam has adequate geotechnical information (including boring logs or cone penetrometer tests) and is well-instrumented with piezometers in the embankment and foundation. Key factors in making this determination include the following:

- Piezometers are properly located and isolated in zones or layers of interest. Piezometric measurements can often be used to evaluate the seepage response and phreatic surface in an embankment dam rather than a model.

- The reservoir level does not fluctuate significantly on an annual basis such that steady-state flow conditions can be assumed. It is more difficult to estimate seepage response from measured data if transient flow conditions exist.
- There is no need to evaluate the embankment dam under a future operating condition (e.g., higher reservoir pool level due to dam raise) or extreme loading condition (e.g., flood pool level). If there are questions about the embankment performance under future operating or extreme loading conditions, a model will be necessary.

Remember: the dam owner often has a limited budget. This is especially important if there is concern that a seepage model may not produce useful results, or if there is a known issue that may make calibration challenging. In general, dam owners prefer to spend their money on tangible items—something they can explain and defend to their board or commission.

Will the Seepage Analysis Accurately Capture the Objective?

Seepage is sensitive to small, localized variations of permeability, defects, anomalies, and fissures that are difficult to identify and model with precision, such that a high level of accuracy can be difficult to achieve in a seepage analysis. A seepage analysis should be considered to give an order of magnitude level of accuracy that is dependent on the estimated permeabilities for the embankment and foundation materials. Therefore, sufficient data on the embankment geometry, internal zoning, foundation contact and stratigraphy, and material properties is necessary for developing effective seepage models. In some cases, additional site exploration and investigation may be required to obtain the necessary data to perform a seepage analysis.

Planning for a Seepage Analysis

Planning for a numerical seepage analysis involves defining the modeling approach and obtaining the minimum data requirements.

Schedule Considerations

A numerical seepage model with relatively simple geometry can be set up and run in a few hours. The more significant effort occurs during the model calibration and verification process. The calibration process requires the modeler to understand the sensitivity of each parameter, and it often takes many tweaks to calibrate the results to both piezometric and observed conditions. The more complex a model's geometry, the more time consuming the calibration process becomes. Often it is necessary for the modeler to run a number of sensitivity analyses to account for uncertainty in material properties. Verifying the results of a model is also time consuming because it requires a detailed understanding of the model inputs and a thorough review of output.

If you are short on time AND are convinced a seepage model is needed, it might be good to subcontract with an expert who is experienced with model setup, calibration, and verification of results.

Special Note on Subcontracting with a Specialist:

There are situations where subcontracting with an expert modeler is the most cost-effective decision:

- **The software is sophisticated and must be used frequently to be time efficient.**
- **The software is expensive to purchase with costly annual maintenance fees.**
- **A transient condition model is difficult to set up and challenging to calibrate.**
- **There is a tight schedule or lack of available personnel.**

One approach is to do the initial legwork (i.e., conceptual model, data acquisition and organization) and then subcontract with an expert to do the modeling, calibration, and interpretation.

Modeling Approach

The first step in performing a numerical seepage analysis involves defining the modeling approach, which includes considerations for the following:

- Dam site and embankment geometry,
- Cross section versus plan view modeling,
- Steady-state versus transient seepage analysis, and
- 2D plane strain versus 3D modeling.

Geometry

The geometry of both the dam site and embankment should be taken into consideration in seepage modeling. In a 2D analysis, cross sections should be selected at locations where critical seepage conditions are expected and seepage results are required. This will typically include the maximum embankment section, at a minimum. In a 3D analysis, an entire dam and surrounding area can be included in the model.

Cross Section versus Plan View Models

Seepage analysis for embankment dams is most commonly modeled using 2D cross sections. However, in some cases a seepage analysis may be modeled in plan view. Although not common, plan view seepage models have been used to evaluate groundwater flow around abutments or cutoff walls. More commonly, plan view seepage analyses are used in the selection and design of dewatering systems or relief wells.

A plan view analysis is intended to model groundwater flow through confined aquifers, so application to unconfined problems must be conducted with caution. Also, it is often more difficult to know the boundary conditions in plan view models. Many dams do not have instrumentation away from the left or right abutments, which makes setting boundary conditions challenging. For more information on plan view models, refer to [2] and [5].

Steady-State versus Transient Analysis

Seepage through an embankment dam can be analyzed under steady-state or transient flow conditions. A steady-state seepage analysis represents the long-term operating condition of a dam. In a steady-state model, internal pore water pressures and flow conditions are computed for a given set of boundary conditions and are assumed to be steady (i.e., unchanging). This condition is typically evaluated with the reservoir assumed to be at the normal operating pool level for a long period of time and is the most commonly analyzed condition.

In a transient seepage analysis, the reservoir boundary condition is not constant. In a transient model, both initial and future boundary conditions must be specified to evaluate how long it takes for the embankment materials to respond to the given set of boundary conditions. Typical transient analysis scenarios include evaluating the wetting front rate through an embankment during the first reservoir fill after construction, the maximum reservoir drawdown rate to meet stability or other requirements of the applicable regulatory agency, the annual pore water pressure regime through an embankment that experiences yearly reservoir fluctuations, and/or the effect of flood loading (i.e., how long it will take to saturate the embankment and reach a steady-state condition). An example of a transient analysis is presented in Figure 5.

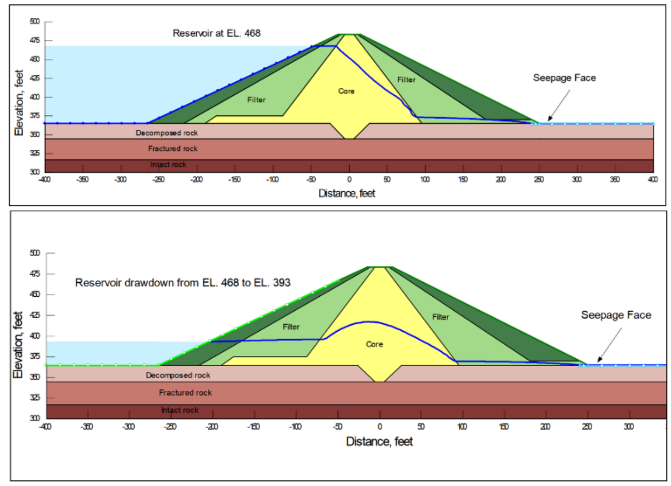


Figure 5: Transient Seepage Analysis for Reservoir Drawdown through a Zoned Embankment (Adapted from [1])

Irrigation reservoirs that experience fluctuations in water levels annually may not be well-represented with a steady-state seepage model. Typically, there is a lag between changes in reservoir water level and corresponding responses in piezometers. When modeling irrigation reservoirs, carefully look at lags in piezometric responses as the reservoir fills, reaches full pool, and starts to draw down. If a significant lag is obvious, it may be necessary to perform a transient seepage analysis to obtain meaningful results.

2D versus 3D Modeling

Historically, seepage analyses have primarily comprised 2D modeling given the complexity of 3D modeling and limited computer programs capable of 3D modeling. However, seepage analyses have recently been expanding into the 3D realm as the benefits of 3D modeling are becoming clearer and computer programs with 3D capability are becoming more prevalent.

While 3D seepage modeling can be valuable under certain conditions, 3D modeling is much more rigorous than 2D and therefore requires careful consideration to ensure 3D modeling is the appropriate approach. 3D modeling can be very time-consuming, requires a greater level of effort and expertise, and is significantly more costly than 2D. Conditions in which 3D seepage modeling becomes beneficial include the following:

- There are significant 3D cross-valley effects along the embankment dam alignment (narrow v-shaped valley profile, irregular/uneven or sloping foundation or abutment surface, pervious foundation or abutment, etc.) – Figure 6.
- A relatively long dam (or levee) that has a convex bend in the embankment. The bend serves as a point where seepage from multiple directions may converge – Figure 7.
- Complex embankment geometry (e.g., discrete seepage control features such as filters/drains, toe drain, relief wells, etc.) and/or foundation geology (e.g., bedrock discontinuities, faults, etc.)

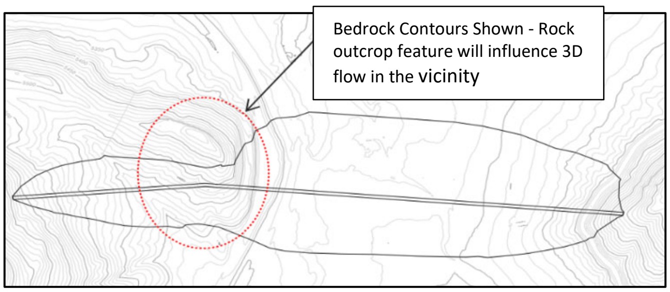


Figure 6: Examples When 3D Seepage Modeling Becomes Beneficial: Irregular Bedrock Foundation

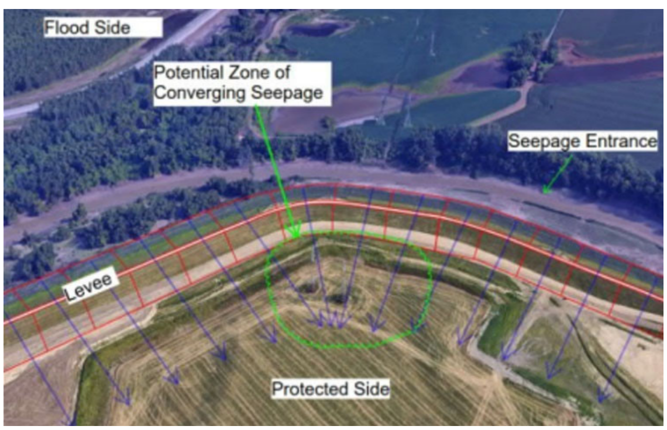


Figure 7: Examples When 3D Seepage Modeling Becomes Beneficial: Convex Bend in Dam Alignment

In all cases, there must be adequate information available to justify the expense of a 3D model. If boundary condition input or geotechnical/geological data are lacking, a 3D model may not be warranted. Note that there often is value to start with a 2D model that is then used to inform a 3D model when 3D flow is an issue.

Minimum Data Requirements

Seepage models can be adjusted based on very little data until the results look like what the modeler hoped they would, but "garbage-in, garbage-out" makes those models a waste of time. The minimum data requirements needed to develop an efficient and reliable seepage model for an embankment dam are summarized in Table 2, along with the purpose of the data and reference documents where the data can be obtained. These minimum data requirements are discussed further below.

Table 2: Minimum Data Requirements for Seepage Modeling

Data Category	Data Requirements	Typical Data Sources
Model Geometry	• Embankment Geometry and Internal Zoning. • Foundation Contact and Stratigraphy.	• Topographic or Lidar Surveys • As-Built Construction Drawings • Design Drawings • Design or Construction Reports • Geological and Geotechnical Investigation Reports
Material Properties	• Embankment and Foundation Permeabilities.	• Geotechnical Investigation and Data Reports • Construction Reports • Published Data
Calibration Data (for Existing Dams)	• Reservoir Levels, Piezometer Data, Weir Flow Data.	• Instrumentation Records • Inspection Reports • Geotechnical Investigation and Data Reports

Model Geometry

Seepage models are typically developed for one or more embankment cross sections along the dam alignment. To develop the model geometry, there must be sufficient available data on the embankment geometry and internal zoning, as well as the foundation contact and stratigraphy. A detailed model geometry will delineate the various embankment zones (core, shells, filters, drains, etc.), foundation layers, and any other seepage control systems (toe drains, low-permeability blankets, cutoff walls, etc.).

The best data source for defining the embankment geometry and internal zoning of an existing embankment dam is typically as-built construction drawings. Ideally, the external geometry (i.e., embankment crest, upstream and downstream slopes, and downstream ground surface) should be defined by a recent topographic or lidar survey. When construction drawings and recent surveys are not available, design drawings and/or design or construction reports can be used for defining the model.

However, the modeler should be aware that the as-built and/or current embankment condition may differ from the design condition. Trust but verify. Information from geological and geotechnical investigation reports can also be valuable in verifying the internal zoning and/or variations in the embankment materials. For a new embankment dam that has yet to be constructed, the embankment geometry and internal zoning is typically defined using design drawings. The design can then be adjusted accordingly based on the results of the analysis.

The best data source for defining the foundation contact and stratigraphy are typically geological and geotechnical investigation reports or pre-construction surveys.

Material Properties

Seepage modeling requires assigning material properties to the embankment and foundation materials. These material properties include saturated permeability, the anisotropy ratio (i.e., the ratio of horizontal to vertical permeability), and some miscellaneous parameters to define the characteristics of flow through unsaturated zones of the embankment. Similar to the model geometry, there must be sufficient available data on the embankment and foundation materials to estimate the material properties. Typically, seepage properties are estimated using data collected from geotechnical investigations and/or published data. Information on the materials may also be available in construction reports.

Permeability values can be estimated from laboratory tests (e.g., constant head, falling head, or flexible wall permeameter), field tests (e.g., borehole soil permeability tests or rock packer tests), published tables of values, and/or empirical equations, which most often relate permeability to material gradation and void ratio. Laboratory and field tests represent the most reliable estimates of permeability.

Published tables of values and empirical equations should be used with caution in regard to the accuracy of the estimated permeability values, and the analyst should consider the specific material for which the empirical correlation is applicable. Most empirical correlations are applicable to granular materials and become unrepresentative for materials with high fines contents.

For seepage models using material properties based only on published tables of values or empirical correlations, users should consider a range of potential permeabilities assigned to the various materials by performing sensitivity analyses to represent uncertainty.

At a minimum, estimating reasonable seepage material properties requires adequate geotechnical/geological data on the embankment and foundation materials, including:

- Soil or rock type,
- Gradations,
- Density,
- Stratification/discontinuities, and
- Compaction procedures.

Table 3 provides ranges of permeability values for a variety of soil types. This is a rough guideline that should only be used to compare permeability estimates as a reality check and not to be used directly in a seepage analysis.

Table 3: Typical Permeability Ranges by Soil Type (Adapted from [7])

Soil Type	Permeability, k (cm/sec)
Clays	$1x10^{-7}$ to $1x10^{-9}$
Very Fine Sands, Silts, Mixtures of Sand, Silt, and Clay	$1x10^{-7}$ to $1x10^{-3}$
Clean Sand, Clean Sand and Gravel Mixtures	$1x10^{-3}$ to 1
Clean Gravel	1 to $1x10^{2}$

For more information on seepage material properties and laboratory and field permeability tests, refer to [1], [3], and [7].

Calibration Data

For existing dams, seepage models should be calibrated. Developing a plan to calibrate the model is an important step in the data review process. Calibrating a seepage model requires adequate records on reservoir levels, piezometer data, and/or weir flow data. Reservoir levels in conjunction with piezometer data can be used to calibrate the phreatic surface or potentiometric surfaces modeled within an embankment. Weir flow data can be used to compare actual measured seepage flow rates with estimated seepage flow rates. Caution should be used when relying on weir flow data to calibrate the seepage model, as weirs can collect water from other groundwater sources, or seepage may not be collected by the weir. Calibration data including reservoir levels, piezometer data, and weir flow data can typically be found in instrumentation records and/or inspection reports. Geotechnical investigation reports can also be reviewed to determine whether groundwater was encountered during test hole explorations. The water levels measured in test holes can also be used to supplement piezometer data for calibrating the modeled phreatic surface.

Interpretation and Communication of Results

After the seepage model is run and results are obtained, it is important to examine the various output features, as these are visual tools that help to interpret and verify the model results. Figure 8 shows an example of seepage model results, and Table 4 summarizes the key output features (specific to GeoStudio's SEEP/W® computer program) and associated tips for checking the validity of the model results. Although the table is specific to the SEEP/W® computer program, the output features and concepts are relatively similar among seepage modeling computer programs.

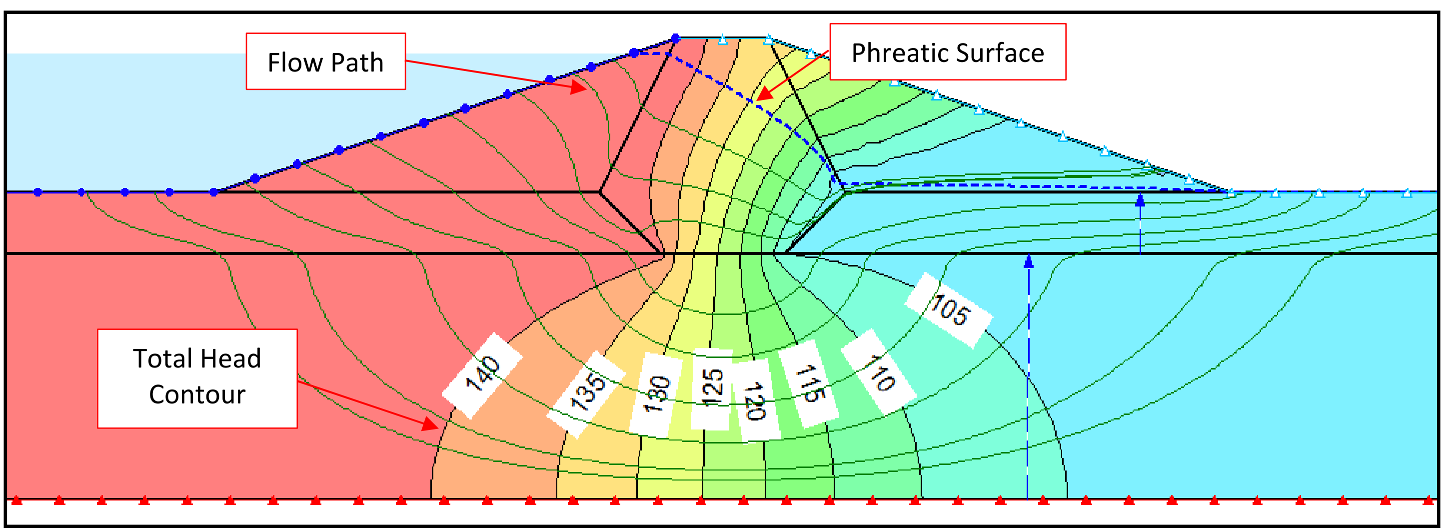
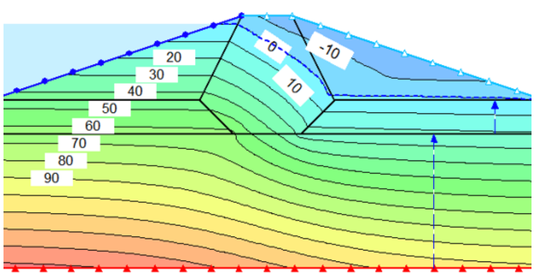
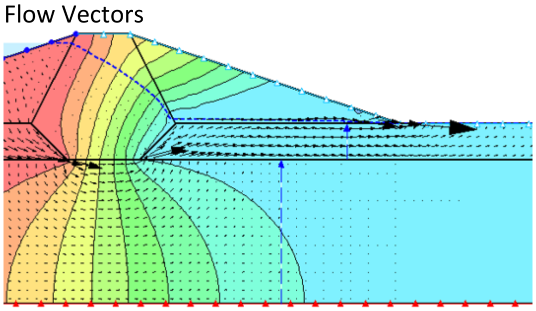
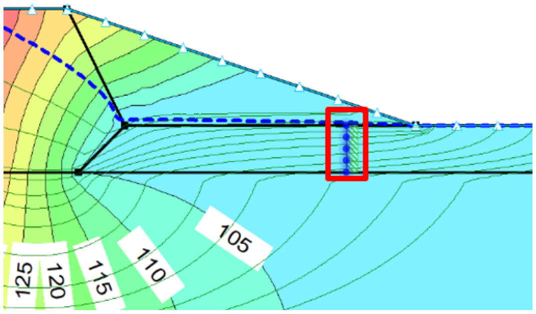
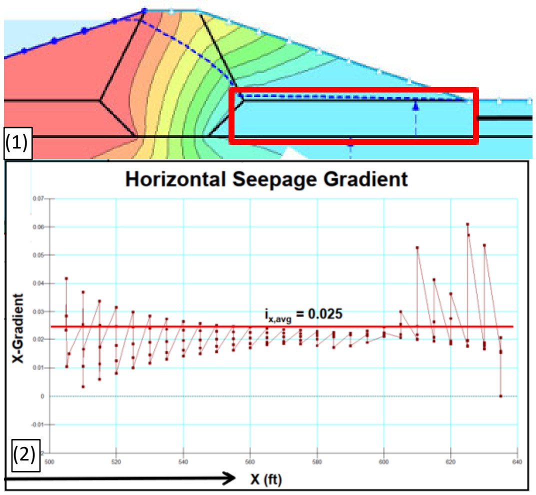


Figure 8: Seepage Model Results Showing Total Head Contours and Flow Paths (Flow Net Simulation)

Table 4: Key Seepage Model Output Features for Interpreting and Verifying the Results

Output Feature	Description	Tips/Considerations
Total Head Contours (1) See Figure 8	• Lines that depict where total head values (i.e., pressure head plus elevation) are the same. • Indicators of the direction of seepage flow.	• Total head contours should decrease from upstream to downstream. • Farthest upstream contour should be equivalent to the elevation of the reservoir level.
Flow Paths (1) See Figure 8	• Lines that individual water particles travel within the flow regime from upstream to downstream under a steady-state condition.	• Flow paths should intersect total head contours at right angles (or at least close to right angles) for homogeneous, isotropic sections. • Flow paths may cross above the phreatic surface since water can flow from the saturated to the unsaturated zone, and vice versa.
Phreatic Surface See Figure 8	• Transition from positive to negative pore water pressures. • Boundary between saturated and unsaturated flow.	• Phreatic surface should be the line of zero pressure. • Undulations or irregularities in the phreatic surface may be an indication that the mesh size needs to be reduced.
Pore Water Pressure Contours 	• Lines that depict where pore water pressure values are the same.	• Pore water pressure contours should become increasingly more positive below the phreatic surface and increasingly more negative above the phreatic surface. • Useful for verifying the model is properly computing the saturated and unsaturated zones. • Areas of high pore water pressure may have a negative impact on slope stability.
Flow Vectors 	• Depict the direction and magnitude of seepage flow.	• Larger vectors (arrows) indicate higher seepage flow velocities. • Smaller vectors indicate lower seepage flow velocities. • Understanding and considering where high and low permeability zones are in the model should help users judge whether the relative flow direction and velocity magnitude make sense.

Output Feature	Description	Tips/Considerations
Flow Sections 	• Defined sections of the model across which unit seepage flow quantity is computed.	• Useful for evaluating the seepage flow rate through a specific material region and/or location of interest.
Horizontal and Vertical Seepage Gradients 	• Change in total head (i.e., head loss) over the length of the flow path. • Two visual options: (1) a diagram can define an area of the model to compute average seepage gradients and/or (2) a graphical representation can depict seepage gradient contours.	• Option (1) is useful for evaluating minimum, maximum, and average horizontal seepage gradients and vertical (exit) gradients through a specific area of interest. • Option (2) is useful as a check to Option (1). The author's preference is to evaluate gradients using Option (1) because point anomalies (i.e., gradient spikes) can be more difficult to define using Option (2). As the contour interval reduces, the gradient point anomaly tends to increase with no set end value.

Note:

(1) Total head contours and flow paths can be used to approximate the flow net. Total head contours are equivalent to equipotential lines in a flow net. However, flow paths are not the same as flow lines in a flow net. Thus, the addition of flow paths to the total head contours can only simulate the flow net. In a flow net, the amount of flow between each flow line (referred to as flow channels) must be equivalent. In a seepage model, flow paths can be drawn at any point within the flow regime of the model such that the flow between flow lines will not always be equivalent.

Conclusion

Seepage modeling can be simple or complex and requires practice and experience. Models will often run without error and produce professional-looking results while still being invalid, so successful runs should not be confused with accurate results. Therefore, vetting errors and understanding the sensitivity of a model to the potential range of each input parameter should be a priority before using the results. This often requires knowledge gained through personal trial and error. Consider seeking assistance and guidance from experienced engineers and modelers. There is always merit in getting a secondary review from an expert, as some regulatory agencies may not have the modeling experience to catch problems.

This article presented an introduction to seepage analysis in embankment dams with a focus on planning and purpose. The reader is further encouraged to read the guidance document developed for the Montana Department of Natural Resources and Conservation, which provides greater detail on the planning aspects of seepage analysis [4].

References

[1] Department of the Interior, Bureau of Reclamation (Reclamation). Design Standards No. 13, Embankment Dams, “Chapter 8: Seepage,” January 2014. https://www.usbr.gov/tsc/techreferences/designstandards-datacollectionguides/finalds-pdfs/DS13-8.pdf

[2] U.S. Department of Agriculture, Natural Resources Conservation Service (NRCS). Soil Mechanics Note No. 7, The Mechanics of Seepage Analyses, October 1979. https://directives.sc.egov.usda.gov/OpenNonWebContent.aspx?content=18467.wba

[3] U.S. Army Corps of Engineers (USACE). Engineer Manual (EM) 1110-2-1901, Seepage Analysis and Control for Dams, April 30, 1993. https://www.publications.usace.army.mil/Portals/76/Publications/EngineerManuals/EM_1110-2-1901.pdf?ver=INPQyoyQDHXDtlSpfJ9fXw%3d%3d

[4] AECOM. Seepage and Slope Stability Modeling for Embankment Dams, A Guidance Document for Planning, Interpreting, Verifying, and Reporting Results, Prepared for State of Montana, Department of Natural Resources and Conservation, November 2021.

[5] GEO-SLOPE International Ltd (GEO-SLOPE). Stability Modeling with SLOPE/W, An Engineering Methodology, November 2012 Edition. http://downloads.geo-slope.com/geostudioresources/8/0/6/books/slope%20modeling.pdf?v=8.0.7.6129

[6] Federal Emergency Management Agency (FEMA). Federal Guidelines for Dam Safety, Emergency Action Planning for Dams, FEMA 64, July 2013. https://www.fema.gov/sites/default/files/2020-08/eap_federal_guidelines_fema_p-64.pdf

[7] Cedergren, H.R. Seepage, Drainage and Flow Nets, Third Edition, John Wiley & Sons, Inc., 1989.

CFD 101 – CFD as a Design Tool: What Does CFD Bring to a Dam and Spillway Project?

By: Frank Lan, Ph.D., PE, AECOM

Introduction

Comprehensive design and engineering assessment of dam-associated hydraulic structures, such as spillways, energy dissipation basins, diversions, intakes, outlets, and fish ladders, requires an understanding of the very complex behavior of rapidly moving water around complex structures. In earlier years, theoretical solutions were derived for simplified geometries; later, scaled physical hydraulic models were constructed in laboratories to examine flow around more complicated structures.

The physical model studies established the empirical basis for developing invaluable design guidelines for many standardized hydraulic structures. However, physical models can be expensive and time-consuming and have difficulties associated with scaling effects to complex real-world applications. Typically, the constraints of a physical model allow only a handful of investigations of final design alternatives, whereas much information can be gained through elegant modeling during multiple alternatives evaluations to support robust engineering analysis.

Computational Fluid Dynamics (CFD) modeling in the dam industry has emerged in recent years to support engineering analysis and refinement through all stages of the design process. Advancements in computational mathematics and computing power since the 1980s, especially in the last two decades, have been proven to produce similar results to physical models at a fraction of the expense at prototype structure scales.

Based on this author's experience with one of the most popular commercial codes for surface flow application, FLOW-3D®, this paper presents the general requirements for a CFD model and discusses many of the benefits as well as some limitations of applying the CFD models in a dam-related project as compared to a conventional physical model and 1D/2D numerical models. Herein, the CFD models refer to three-dimensional CFD models that distinguish them from the traditional one-dimensional or two-dimensional hydrodynamic models such as the popular and powerful HEC-RAS developed by the U.S. Army Corps of Engineers.

This introduction to CFD in dam- and spillway-related applications targets engineers and project managers interested in learning the fundamentals of CFD modeling for dam and spillway hydraulic structures. This article discusses the reasons for conducting a CFD analysis, some of the basic requirements for creating a complete CFD model, the benefits and drawbacks of running a CFD model, and some of the challenges that are still facing the industry. This article also presents some example projects for a range of applications, from simple spillway capacity calculations to more complex structures such as stepped spillways, intakes, converging spillways, piano key and labyrinth weirs, stilling basin, fishway design, and others under various complicated flow conditions.

What Is CFD?

Computational Fluid Dynamics, commonly called CFD, employs numerical analyses and methods to approximate fluid flow behavior governed by the partial-differential Navier-Stokes equations. Computers are used to automate the labor-intensive iterative calculations required to evaluate the behavior of the fluid flow (e.g., velocity and pressure) while accounting for physical properties of the fluid and entrained scalars (e.g., density and viscosity, air, sediment, temperature).

The numerical algorithms underlying CFD models require a three-dimensional grid or mesh (i.e., solution space) to discretize the Navier-Stokes equations (conservation of mass and momentum). Models produce solutions for the fluid behavior at each intersection point of the grid and at each point in time of the overall simulation. The fluid behavior is computed throughout the solution space with algorithms typically defined as finite difference, finite volume, or finite element, each of which having associated pros and cons. Further, numerical scheme can resolve the interactive motion of a fluid through space and time and complex subscale behavior. The book, "Verification and Validation of 3D Free-surface Flow Models" by Wang et al. (2008), published by ASCE [2], provides a detailed discussion of theory, the equations of motion, numerical methods and techniques, discretization methods, and turbulence models.

History of CFD

CFD has existed in some form for more than a century, yet its acceptance as a design tool in dam- and spillway-associated projects did not flourish until the 1990s. The history of CFD can be summarized as [1, 3, 5, 6, 7, 9, 10]:

- **Pre-1910:** Improvements on mathematical models and numerical methods.
- **1910s–1930s:** Integration of models and methods to generate numerical solutions based on hand calculations [3]. Earliest numerical solution for flow past a cylinder in 1933 [8]
- **1940s–1950s:** Transition to computer-based calculations with early computers (ENIAC) [5]. Solution for flow around a cylinder by Kawaguti with a mechanical desk calculator in 1953 [10]. **196s–1970s:** Initial study using computers to model fluid flow based on the Navier-Stokes equations by Los Alamos National Lab, US, led by Francis H. Harlow. A variety of numerical methods to simulate transient 2D fluid flows were introduced. First scientific paper about computational analysis of 3D bodies, "Calculation of potential flow about arbitrary bodies," published by Hess and Smith in 1967 [7]. Commercial codes with contributions of various methods such as k-ε turbulence model, Arbitrary Lagrangian-Eulerian, SIMPLE algorithm, were generated, which are all still broadly used.
- **1980s–1990s:** Codes generated by Boeing, NASA and some have started to use for several purposes such as determining fluid behavior around submarines, surface ships, automobiles, helicopters, and aircraft [6]. Improvement of accurate solutions of transonic flows in the three-dimensional case by Jameson et. al [9]. Commercial codes started to be implement through both academia and industry [9].
- **1990–Present:** Advancement in computing make worldwide usage of CFD possible in every sector, and its application in Dam and Spillway design becomes popular.

Governing Equations

The governing equations of the fluid motion are based on the following fundamental conservation laws of mass, momentum, and energy:

1. Conservation of Mass: Continuity Equation
2. Conservation of Momentum (widely known as Navier-Stokes equations): Newton's Second Law
3. Conservation of Energy: First Law of Thermodynamics or Energy Equation

These principles state that mass, momentum, and energy are stable constants within a closed system and must be conserved. They can be respectively expressed using the condensed partial differential equations (PDEs) below.

The mathematical governing PDEs are presented below only to show the complexity of the equations that CFD can solve as an introduction to CFD. Further reading to understand the theory and the numerical schemes for CFD would be greatly beneficial to building a successful CFD model and providing solutions to the design.

The PDE for the **Conservation of Mass** is specified as [13] is:

$$\frac{D\rho}{Dt} + \rho(\nabla \cdot \vec{v}) = 0 \qquad (1)$$

where ρ is the density, $\vec{v}$ the velocity and $\vec{\nabla}$ the gradient operator.

$$\vec{\nabla} = \vec{i}\frac{\partial}{\partial x} + \vec{j}\frac{\partial}{\partial y} + \vec{k}\frac{\partial}{\partial z} \qquad (2)$$

If the density is constant, the flow is assumed to be incompressible, and the continuity equation reduces to:

$$\frac{D\rho}{Dt} = 0 \;\rightarrow\; \nabla \cdot \vec{v} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (3)$$

Conservation of Momentum is generally referred to as the Navier-Stokes equation and is given by:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\tau) + \rho\vec{f} \qquad (4)$$

where p is static pressure, τ is viscous stress tensor and $\rho\vec{f}$ is the gravitational force per unit volume.

Viscous stress tensor τ can be specified as below in accordance with Stokes' Hypothesis:

$$\tau_{ij} = \mu\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_x} + \frac{2}{3}(\nabla \cdot \vec{v})\delta_{ij} \qquad (5)$$

If the fluid is assumed to be incompressible with constant viscosity coefficient, μ is assumed constant, and the Navier-Stokes equation simplifies to:

$$\rho\frac{D\vec{v}}{Dt} = -\nabla p + \mu\nabla^2\vec{v} + \rho\vec{f} \qquad (6)$$

Conservation of Energy is the first law of thermodynamics, which states that the sum of the work and heat added to the system will result in the increase of energy in the system:

$$dE_t = dQ + dW \quad (7)$$

where dQ is the heat added to the system, dW is the work done on the system, and dE_t is the increment in the total energy of the system. One common type of energy equation is:

$$\rho\left[\frac{\partial h}{\partial t} + \nabla \cdot (h\vec{v})\right] = \frac{\partial p}{\partial t} + \nabla \cdot (k\nabla T) + \phi \quad (8)$$

where T is the temperature and Φ is the heat source. In dam and spillway applications, conservation of energy is rarely applied, assuming that temperature maintains as a constant.

What Can a CFD Model Do?

CFD modeling represents the best available approximations of hydraulic conditions at existing or hypothetical dam configurations apart from direct field measurements. CFD modeling not only captures flow behavior in typical engineering applications (e.g., 1D, 2D) but is the only solution available for many engineering problems where analytical solutions do not exist or where physical experimental approaches are not manageable. This type of modeling has been proven cost-effective, time-efficient, and reliable for a variety of science and engineering applications.

CFD modeling has a wide range of application in water resources and environments:

- River hydraulics
- Outfalls and effluents
- Diversions and in-stream structures
- Sediment transport, scour and deposition in rivers, lakes, and reservoirs
- Fish ladder, naturalized fishway, invasive species barriers and hazardous hydraulics
- Spillways and stilling basins
- Inlets and outlets
- Dam breaches
- Landslide-induced wave analyses
- Tailrace hydraulics
- Wave analyses
- Overtopping and wave run up analyses
- Coastal structures
- Inland waterways
- Drop structures
- Combined sewer overflows
- Tunnel flow analyses
- Flow splitting/bifurcations
- Pump stations
- Clarifiers
- Aeration analyses
- Etc.

In dam-associated projects, CFD modeling is commonly used for:

- Developing outflow rating curves for structures such as spillways and lower outlets
- Evaluating fishway hydraulics and fish entrainment exclusion hydraulics
- Determining diversion dam hydraulics
- Evaluating hydraulic conditions at inlets and outlets to identify adverse flow conditions such as flow separation, vortices, cavitation potentials, head losses, etc.
- Calculating water surface profiles down spillway chutes to develop free board requirements
- Identifying cross waves
- Evaluating performance of energy dissipation basins
- Determining dynamic hydraulic loading on structures
- Evaluating scour potential downstream of hydraulic control structures
- Evaluating sedimentation potential at intakes and identifying mitigation measures
- Evaluating hydraulics during the opening and closing of gates
- Developing operational curves for efficient management of structures
- Cost optimization of structure design, operations management, and maintenance strategies

As a design tool, CFD modeling can effectively:

- Provide preliminary design analysis. A new project or a rehabilitation design may benefit from a CFD model that can provide detailed information regarding a proposed design when this information cannot be obtained from design guidelines, simplified 1D or 2D models.
- Support design decisions: CFD modeling is commonly used during all phases of design, from investigating preliminary geometries and features to optimizing acceptable designs to analyzing

specific circumstance or design modifications that may arise during construction.

- Coupled with a physical model (either in series or parallel), known as hybrid or composite modeling, to be used to set up accurate boundary conditions for physical modeling.
- Provide a tool to direct operation and maintenance decision. For example, a CFD model may be used effectively to analyze various operation schemes for a spillway with multiple gates without requiring actual field testing and loss of water storage.
- Identify hazardous hydraulic structures flow range
- Predict hydraulic impacts of sediment deposition on structure performance and assist with formulating a maintenance plan or rehabilitation scheme.
- Serve as a research tool to enhance understanding. CFD modeling compliments physical models and provides new insights to hydraulic structures.

CFD Models vs. Other Models

Benefits of a CFD Model over 1D and 2D Numerical Models

Modeling hydraulic structures requires selecting the right tool for the job at hand. CFD modeling represents the highest level of sophistication compared to 1D or 2D models, although lesser-dimensional models can provide similar answers in less complex flow environments. CFD modeling maintains distinct advantages over more simplistic methods:

- A CFD model can be applied for circumstances that a 1D or 2D may not be applicable.
- A CFD model provides a more accurate and detailed answer, especially in complex flows.
- The more accurate result of a CFD model could reduce optimistic or "overly conservative" estimates from simplified 1D or 2D numerical models and could reduce project construction costs and project risks.
- CFD models require fewer assumptions and calibration parameters than 1D or 2D models.
- Lastly, the setup times and costs of using a CFD model over a conventional 1D or 2D model are nearly the same, depending on the complexity of the problem. The only major difference is that the CFD model requires much longer run times, typically in days for each run, depending on the grid size, time step needed, and complexity of the problem.

Benefits of a CFD Model over a Physical Model

- CFD model setup time has become easier and less costly (less time consuming) with the help of the graphical user interface that comes with the CFD programs. Typical CFD models, even ones with complicated shapes and multiple structures, can be constructed in a few hours to at most a couple of days by an experienced modeler. This can be a significant cost savings compared to the construction of a physical hydraulic model.
- Most physical models will require a setup time of at least a couple of weeks to a couple of months depending on the scale of the project and the complexity of the structures to be tested. In the past few years, the cost of constructing and running a CFD model for multiple simulations versus the construction and testing of structures in a physical model has typically been about half to significantly less.
- While a physical model is limited by space in the laboratory and sometimes it is difficult to set up a headbox to create the proper entrance condition, a CFD model can completely eliminate such a shortcoming and simulate approach conditions much more accurately.
- A structure design can be easily modified in the model to correct undesirable flow conditions or increase structure efficiency, while it may take a few weeks to make such changes in the laboratory (model as well as instrumentation) to modify a structure design in a physical model.
- Visual images in a CDF model easily show design hydraulic parameters (e.g., velocity, depth), flow patterns, and flow direction for better results that can be more easily communicated.
- Hydraulic results can be obtained at every point within the CDF computational model domain, whereas a finite number of parameters can be measured in a physical model.
- Human errors and positioning tolerance errors for instrumentation and laboratory data collection are eliminated in a CDF model, as is the potential for interaction of instrumentation with flow.

- CFD is based on a prototype, which effectively eliminates the scaling problems facing a physical model.

CFD Model Limitations

All modeling tools, both numerical and physical, have limitations. For CFD modeling, some limitations include the following:

- CFD modeling requires a high level of expertise, both in fluid mechanics and in computing technology. CFD modelers without proper training and experience can produce erroneous results. A model will only give results that reflect what the modeler provides (model setup and input values).
- Resolving highly complex flow phenomena, such as the flow on stepped spillways and stilling basins, are still an area of CFD research; some specific applications are still limited in their accuracy.
- Codes that subdivide the domain into many cells for computing are sensitive to the mesh size and structure. A mesh convergence study (also known as a grid convergence study) is needed to document that the selected mesh is appropriate for the purpose of the model.
- Although multiple simulations may be performed in parallel, specific physical problems (e.g., modeling a reservoir and spillway for the 72-hour rainfall event) can take a long time to resolve using CFD modeling, and computational limits still restrict model spatial and time scales due to the time and computational costs.
- Field or physical model data may not be available to calibrate and evaluate the accuracy of a CFD model. In such cases, there may be a wide latitude in the application of the theory and corresponding model features, which increases model uncertainties. This is particularly true for air entrainment since data from prototypes used to verify the air entrainment modules in CFD are still limited and verification has not been thoroughly conducted, especially in spillways and stilling basins with high velocity and high air entrainment capability.
- If many hydraulic conditions (e.g., flow rates or a long duration of a hydrograph) must be evaluated, it may be unrealistic to perform this study with a CFD model because it may take weeks or months when the same data can be obtained within hours using a physical model. However, this is largely dependent on the quantity of data collected in the physical model for each discharge.
- Simulation of material movement, such as sediments, debris, machinery, local scour, slurry flows, and embankment failures, is possible with a CFD model, yet exact results rely on model calibrations.
- CFD models designed around a specific problem or question may not include or be capable of capturing flow instabilities that may be of interest to a project team. For example, unexpected flow surging, nappe oscillations, or structural vibrations may not be captured in a numerical model but could appear in a physical model.
- Simulating splash and spray can be computationally expensive to resolve in a CFD model. Also, depending on the model, viscosity effects may reduce the amount of splash and spray predicted by a CFD model.
- Observers tend to believe and place confidence in CFD output, which can be a limitation or challenge if the compelling images present erroneous information. Apparent computer veracity is dangerous and can be abused.
- The combination of grid size and time step needed can make run lengths unacceptably long.
- The project team normally does not have the opportunity to interact with community members and receive immediate feedback. Although CFD modeling graphics can be compelling and convincing, observing information on a screen is a less engaging experience than a field test or observing a physical model.

When Should a CFD Be Used?

The use of a CFD model for a design will be highly contingent on the complexity of the problem and project budget (especially when considering CFD versus a physical model). Many advantages of a CFD model make it tempting to apply to projects for which a simpler approach could provide the data for the design. For standard designs, the well-developed procedures in many references from the U.S. Army Corps of Engineers, U.S. Bureau of Reclamation, Federal Emergency Management Agency, Federal Highway Administration, etc. would still be the primary choices since they are well tested and validated in the laboratory and field.

CFD modeling should only be considered when the design can't be accomplished using well-documented procedures and methodologies.

The choice between CFD modeling and the more traditional 1D (such as SpillwayPro) or 2D (such as HEC-RAS) will likely be more contingent on the cost of the software (if using commercial CFD) and the ability of the engineers to develop a CFD model efficiently. If there is reasonable doubt that a 1D or 2D model would be able to resolve the hydraulics, a CFD model should be considered upfront since CFD modeling can be done with equivalent or nominally higher costs compared to a 1D or 2D model. Skilled and experienced CFD model users can drive the cost of simulations significantly lower while achieving the goals of the modeling.

A CFD model should be considered when a physical model is deemed necessary for the design. A CFD model could be utilized during the initial alternatives screening and evaluation prior to physical modeling of a final design configuration. This process would aid in finding an optimal design to be tested in the physical model and provide some valuable information for setting up the physical model, which is typically limited by laboratory space and scale. A CFD model could also be completed in parallel with the physical model to help identify issues with either of the models and provide calibration comparison between the two approaches.

Project Examples

The following are some example projects illustrating the benefits of using a CFD model.

Example 1 (Figures 1 and 2): A CFD model was used to help design a side-channel spillway that is limited in space with unfavorable approach conditions, and the control weir could experience significant submergence from the downstream conduit. The calculation would be nearly impossible using empirical methods from references such as Design of Small Dams (USBR, 1987). 1D and 2D models are not applicable for such truly three-dimensional flow conditions.

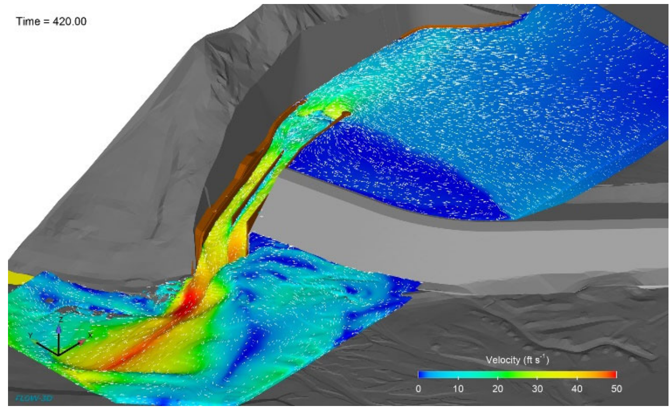


Figure 1: CFD was used to help design a side-channel spillway that could be significantly impacted by the downstream conduit.

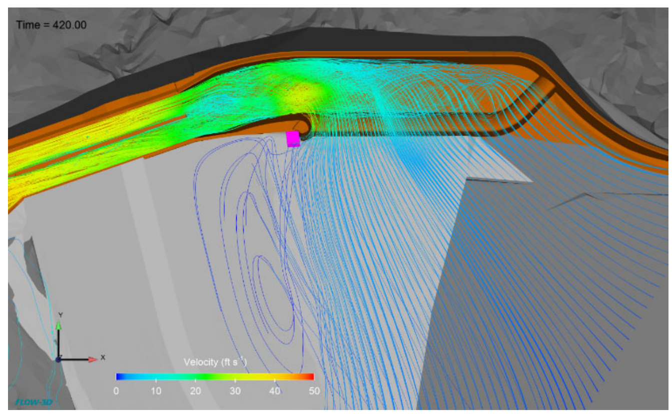


Figure 2: Streamlines showing flow entering a side-channel spillway and through a conduit.

Example 2 (Figure 3): A CFD model was used to help design a new labyrinth spillway that is limited by site geological condition and angled approach flow conditions, with the knowledge that the efficiency of the labyrinth spillway is strongly affected by the attacking angle of the approach flow.

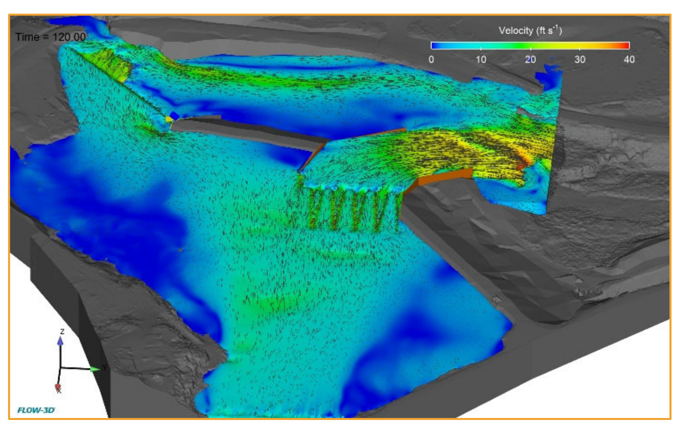


Figure 3: CFD model to screen the spillway configuration to pass the design flood.

Example 3 (Figure 4): A CFD model was used to help develop an outflow rating curve for overtopping of the dam and pertinent structures (spillways and fish inlets), and later to evaluate the scour potential under the Inflow Design Flood (IDF). Hand calculations would require many assumptions, and 1D and 2D models are not applicable for such truly three-dimensional flow conditions.

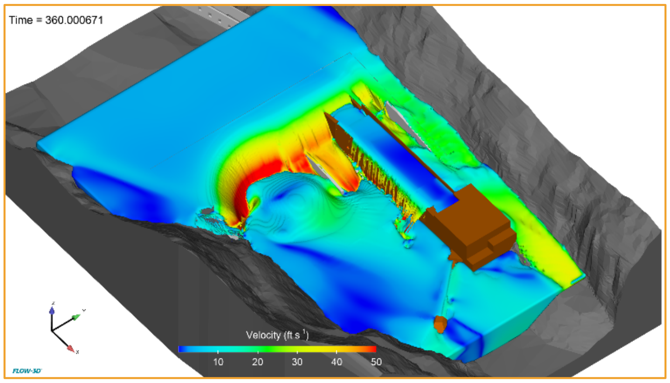


Figure 4: CFD model to evaluate dam overtopping hydraulics, especially the scour potential downstream of the dam.

Example 4 (Figure 5): For a dam with a complex entrance condition, a CFD model was used to evaluate an alternative to significantly increase the capacity of the spillway and subsequently evaluate hydraulics down the curved and converging spillway chute and at the terminal. The CFD model was able to aid in the design of a curved labyrinth configuration that met the design criteria and saved millions of dollars in construction costs when compared to the initial design. The final configuration was later tested in a physical model that produced similar results to the CFD results.

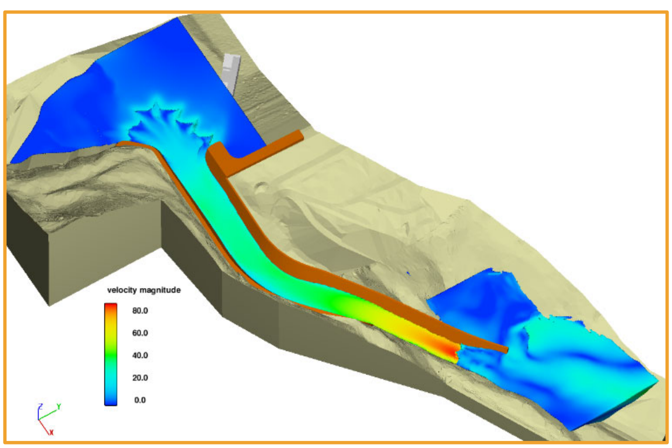


Figure 5: CFD model to help optimize a spillway configuration that could save millions for the project.

Example 5 (Figure 6): A CFD model was used to evaluate the hydraulics at a diversion structure to determine diversion efficiency at different flood conditions and estimate dynamic loading on the gates and bridges from wave effects, as well as provide hydraulic parameters for the design of a stable riffle downstream of the diversion.

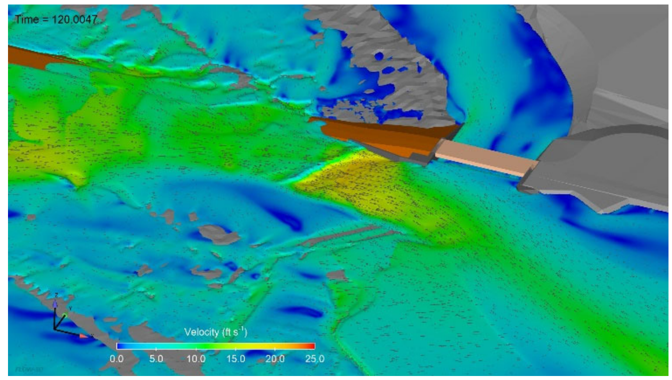


Figure 6: CFD model to evaluate hydraulics at a gated diversion structure.

Example 6 (Figure 7): For a dam with a complex entrance condition, a CFD model was used to develop an outflow rating curve for a proposed tunnel outlet as well as identify any adverse hydraulic conditions, especially vortices at the entrance, associated with the design.

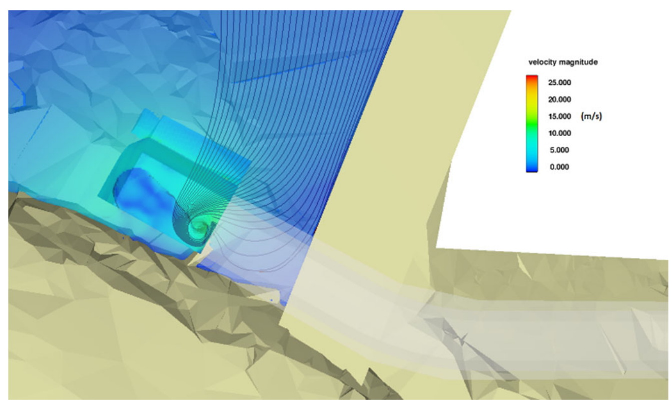


Figure 7: CFD model to evaluate hydraulics, especially vortices at the entrance to a tunnel outlet.

Example 7 (Figure 8): A simple CFD model was used to evaluate the hydraulic performance down a stepped spillway and stilling basin. This evaluation could've also been completed using empirical equations from published design literature.

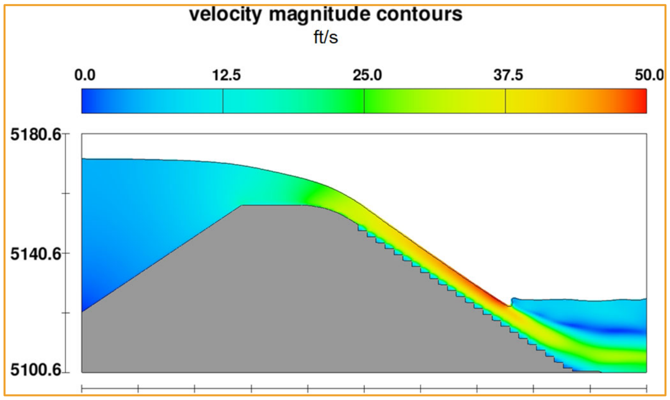


Figure 8: CFD model to evaluate stepped spillway chute hydraulics.

How Is a CFD Model Developed and Used?

Model setup for a CFD analysis can be challenging given the level of simulation space, number of parameters, and numerical solver configuration that is required. This process has been optimized with user-friendly Graphical User Interfaces (GUIs) that come with commercial programs; however, they can remain time-consuming with open-source software. In the example of the commercial CFD package, FLOW-3D® for example, an external graphical design program such as AutoCAD or others will normally be used to construct the solids of the representative topographic or structural features to be modeled. These solids are generally saved as stereolithography (STL) files. From the solids built, a mesh within the pre-defined computation domain will be generated within the FLOW-3D® GUI. This GUI will also facilitate the setup of boundary conditions, initial conditions, and other related flow and roughness properties for the model, and has been streamlined for the CFD engineering workflow.

In general, an experienced modeler can set up a complicated model in less than a day. Additional time may be required to build the topographic and structural solids outside of the CFD models. The GUI normally has the capability to test the model setup to identify some potential problems in the model execution, urging the users to make modification to the setup until it meets tolerances and specifications for smooth and efficient simulations.

One example of a simple model setup that has saved time and project costs for an existing physical model was for a proposed labyrinth spillway for a dam upgrade project. The purpose of the simple CFD model was to investigate whether the proposed structure (the wall heights would be the world's tallest) would function as designed and whether it was worth the effort and cost to make significant alterations to the physical model (this would require removal of large portions of the fixed mortar lining and repositioning of measurement devices). Once the topographic and structure solids were available, the CFD model was set up in half a day, and the numerical result for one design flow discharge was obtained in less than 24 hours computing time, which showed that the option would be worth pursuing. The effort to alter the physical model was estimated at two weeks, compared to the simplified CFD model that was constructed and run in a day.

What Does It Take to Run a CFD Model?

Experienced and Knowledgeable Engineers

CFD simulations require a high level of expertise and thus well-trained engineers and teams who are able to:

- Understand the hydraulics, algorithms, and mathematics behind it, not just the tools
- Build models that will run efficiently and do not require re-runs of AutoCAD and GUI
- The ability to use other methods to reasonably validate the CFD model results
- The ability to interpret and communicate the results
- Computer science and programming background

Software

A suite of open source and commercial CFD codes are available and applicable for water and environmental applications, in particular in dam- and spillway-related projects. Some of the most popular codes include:

- FLOW-3D® – a commercialized CFD product that is based on the finite-volume method developed by Flow Science, Inc. While the model can be applied to many fluid situations, it is particularly robust in dealing with free surface flow that is seen in dam-associated hydraulic structures. The program uses proprietary volume-of-fluid (VOF) techniques to track the free surface so that the free surface may be predicted accurately. The software also features a multi-physics suite of scenarios that includes advanced physical models, turbulence, and a variety of flow types. FLOW-3D uses a simple grid of rectangular elements, so it has the advantages of ease of generation, regularity for improved numerical accuracy, and minimal memory storage requirements. Geometry is then defined within the grid by computing the fractional face areas and fractional volumes of each element that is blocked by obstacles. The equations of motion are then solved based on a finite difference technique.

- Ansys Fluent® – a commercialized CFD product that is based on a finite volume/finite-element method developed by Ansys, Inc. Ansys Fluent can be used for all types of fluid process as well as fluid-structure multi-physics interaction.
- Ansys CFX® – another commercialized CFD product being offered by Ansys, Inc., with similar capability in as Fluent.
- OpenFOAM – a Linux/Unix-based open source CFD product that is available free of charge from The OpenFOAM Foundation Ltd. in England. The software also comes equipped with finite element analysis (FEA) software and computer-aided engineering (CAE) software capabilities. It allows the users to create 3D models based on 2D CAD drawings and replicate the effects that airflow, heat, and water will have on the performance of the designs.
- Star-CD/CCM+® – a commercial CFD product that is distributed by CD-adapco of Melville, NY.

While these CFD programs offer similar capability in modeling flows that are generally required for dam-associated projects, they differ in methods for setting up the model runs, simulation efficiency, and engineer workflow. In general, models based on the finite difference discretization techniques such as FLOW-3D® require the least time to set up, while the models based on the finite-element discretization techniques such as Fluent may take more time, especially in the solution space generation and alteration of subsequent simulations (restart runs). As the geometry of the structure changes, the former technique may not need to re-define the meshes and could potentially save more time. The open source-based model OpenFOAM would require the most time setting up the model based on the author's experience.

Software and Computing Cost

The upfront cost of owning a CFD program license from any of the commercial vendors is generally in the same range from \$50k to \$100k, depending on the configuration of the license.

All CFD programs require high-end workstations for large models that are generally necessary from dam-associated projects. These workstations should have a large number of physical cores (more than 16) and sufficient RAM (more than 32 GB) and storage (recommended more than 2 TB). Specifics of computer builds are program specific, as different programs parallelize more or less efficiently with specific configurations and/or hardware platforms. Cloud computing is also available; however, the cost could be significantly higher for models that require extended computational time since the cost is based on the computer cores to be used and the run time of the models. A single run in the cloud could cost from a couple of hundred dollars to up to several thousand dollars or higher and sometimes can be dependent on time of day.

For small projects that may require CFD modeling on a short-term basis, the easiest approach would be to use the cloud service or purchase short-term licenses. The modeling task could be completed for a few thousand dollars or less.

Open-source code OpenFOAM offers an attractive option to save cost for running CFD models for the long run since there is no license fee to incur. However, the cost of preparing the model and post-processing results from the model could be significantly higher due to the technical background demanded of its users. This cost would go down for experienced and frequent users and will likely decrease in the future as the CFD community grows and open-source tools become more user-friendly and adopted.

Conclusion

With the advancements in computational mathematics and computing power in recent years, CFD modeling has become a powerful and invaluable tool in dam-associated hydraulic structure designs and evaluations. It has the capability of producing reliable results with some cost efficiency when compared to the traditional 1D or 2D models for simple conditions and also for more complex conditions where 1D or 2D models are not suitable. CFD modeling also has the advantage over physical models in terms of cost and efficiency. Knowing that it still has limitations, especially in high-turbulence flow situations with significant air entrainment, CFD modeling could be well suited for finalizing designs that could then be further tested and confirmed in a physical model study. This could potentially result in significant savings in the design process.

The successful application of CFD modeling to a dam-associated project requires the modelers' understanding of the complex hydraulics behind the design, the ability to develop efficient models and know what the model is doing, and the ability to interpret and communicate the results to their clients and all the stakeholders. These abilities to run, understand, and communicate results of CFD models will be invaluable to the dam safety industry.

References

[1] https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/

[2] Wang, S. Y.; Patrick J. Roache, Ph.D.; Richard A. Schmalz Jr., Ph.D.; Yafei Jia, Ph.D.; and Peter E. Smith, Ph.D., P.E. (Editors), Verification and Validation of 3D Free-Surface Flow Models, sponsored by Task Committee on 3D Free-Surface Flow Model Verification and Validation, Environmental and Water Resources Institute (EWR) of the American Society of Civil Engineers, 2008.

[3] Hunt (1997). "Lewis Fry Richardson and his contributions to mathematics, meteorology, and models of conflict". Annual Review of Fluid Mechanics. 30 (1): xiii–xxxvi.

[4] https://www.metlink.org/wp-content/uploads/2020/11/forecast_factory_weather_feb2011.pdf

[5] https://www.bakker.org/cfmbook.pdf

[6] https://en.wikipedia.org/wiki/Computational_fluid_dynamics

[7] http://www.sciencedirect.com/science/article/pii/0376042167900036

[8] Thom, A., 1933. "The Flow Past Circular Cylinders at Low Speeds", Proc. Royal Society, A141, pp.651-666, London.

[9] Jameson, A., Schmidt, W. and Turkel, E., 1981. "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes", AIAA 14th Fluid and Plasma Dynamic Conference, June 23-25, 1981, Palo Alto, CA.

[10] M. Kawaguti, "Numerical Solution of the NS Equations for the Flow Around a Circular Cylinder at Reynolds Number 40", 1953, *Journal of Phy. Soc.* Japan, vol. 8, pp. 747-757.

[11] *Frank M. White, Viscous Fluid Flow,* McGraw-Hill Mechanical Engineering, 3rd Edition, ISBN-10: 0072402318.

[12] Flow Science, Inc. Hydraulics Training Class, FLOW-3D.

[13] Anderson, John D. Jr, 1995. *Computational Fluid Dynamics, the Basics with Applications,* McGraw-Hill, Inc., New York.