United States Department of the Interior

BUREAU OF RECLAMATION
PO Box 25007
Denver, CO 80225-0007

IN REPLY REFER TO:
86-68110
2.2.4.22

VIA ELECTRONIC MAIL ONLY

MEMORANDUM

To: Technology Development Program Manager, Dam Safety Office
            Attn: 84-44000(LKrosley)

From: Lan Nguyen, Civil (Structural) Engineer
            Waterways & Concrete Dams Group 1 (86-68110)

Subject: Transmittal of Dam Safety Technology Development Report DSO-2018-09- Guide for Analysis of Concrete Dam Structures using Finite Element Methods.

Attached for your use is the DSO-2018-09-Guide for Analysis of Concrete Dam Structures using Finite Element Methods report, which has been prepared by the Technical Service Center at the request of the Dam Safety Office. The report will be available in Adobe Acrobat Format on the Dam Safety website and will be loaded into DSDAMS.

If you have any questions, please contact me at 303-445-2128 or via email at LTNguyen@usbr.gov.

cc: DSDAMs
            Archives
            86-68110 (Nguyen, Dominic, Salamon)
            86-68313 (Krest)
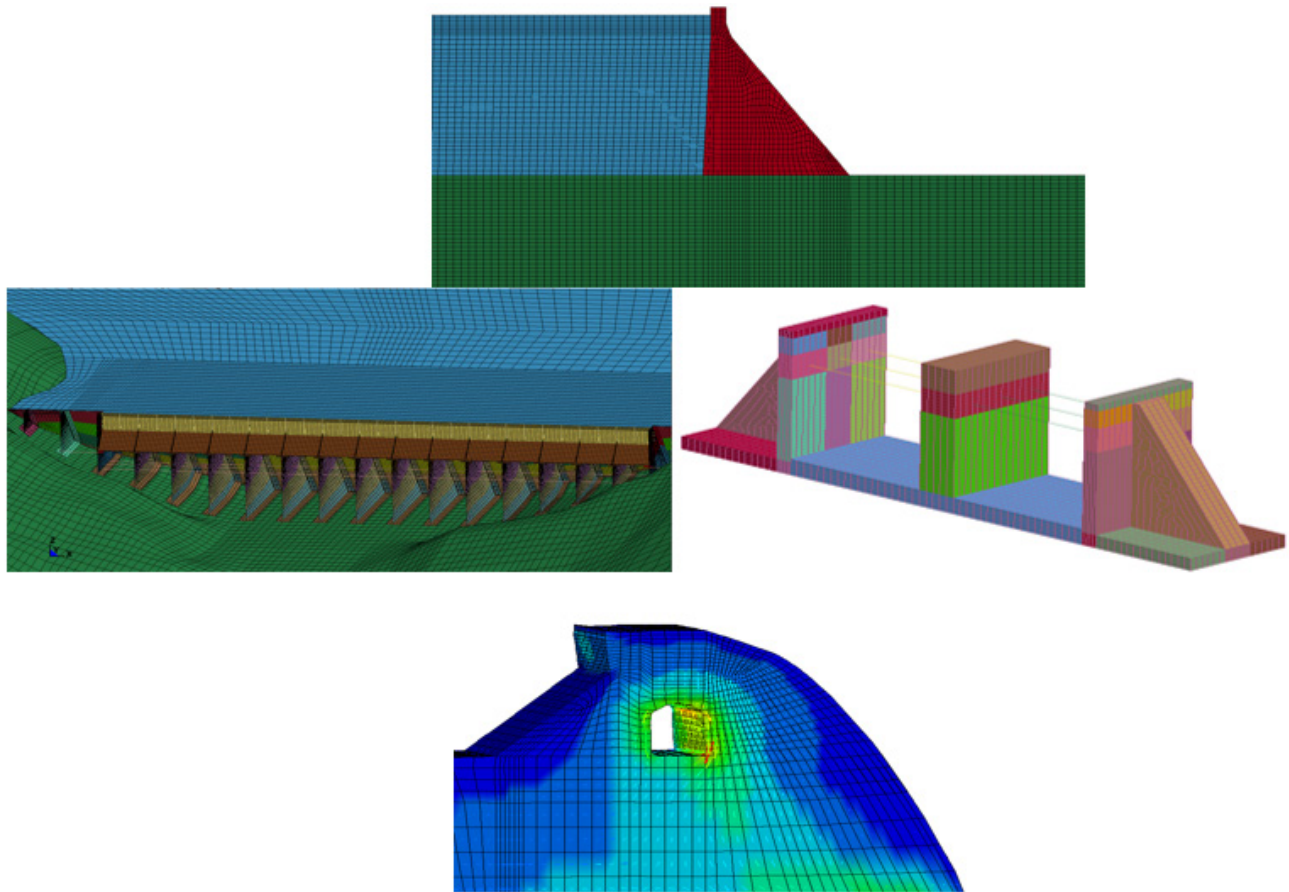   86-68530 (Bartojay)

RECLAMATION
Managing Water in the West

Guide for Analysis of Concrete Dam Structures using Finite Element Methods

DSO-2018-09

Dam Safety Technology Development Program

Mission Statement

Protecting America's Great Outdoors and Powering Our Future.

The Department of the Interior protects and manages the Nation's natural resources and cultural heritage; provides scientific and other information about those resources; and honors its trust responsibilities or special commitments to American Indians, Alaska Natives, and affiliated island communities.

<table>
<tr><td colspan="2" align="center">REPORT DOCUMENTATION PAGE</td><td>Form Approved
OMB No. 0704-0188</td></tr>
</table>

T1. REPORT DATE: MARCH 2018	**T2. REPORT TYPE:** TECHNICAL	**T3. DATES COVERED** December 2015-March 2018

T4. TITLE AND SUBTITLE Guide for Analysis of Concrete Dam Structures using Finite Element Methods	**5a. CONTRACT NUMBER**
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) Lan Nguyen, Ph.D., P.E, Denver, CO (BOR, 303 445 2128) Jerzy Salamon, Ph.D., P.E, Denver, CO (BOR, 303 445 3219)	**5d. PROJECT NUMBER** DSO-2018-09
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER CDAG7

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Department of the Interior, Bureau of Reclamation Technical Service Center, Denver Federal Center PO Box 25007 (86-68110) Denver, CO 80225-0007	**8. PERFORMING ORGANIZATION REPORT NUMBER**
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Department of the Interior, Bureau of Reclamation Dam Safety Office, Denver Federal Center PO Box 25007 (84-44000) Denver, CO 80225-0007	**10. SPONSOR/MONITOR'S ACRONYM(S)** **BOR/USBR:** Bureau of Reclamation **DOI:** Department of the Interior **DSO:** Dam Safety Office
	11. SPONSOR/MONITOR'S REPORT NUMBER(S) DSO-2018-09

12. DISTRIBUTION / AVAILABILITY STATEMENT

13. SUPPLEMENTARY NOTES

14. ABSTRACT This report summaries our work for the Dam Safety funded research project "Guide for Concrete Dam Analysis Using Finite Element Method". Here we discuss fundamental topics such as the mathematical formulation in term of both field partial differential equations and variational principles, boundary conditions for dam and fluid-dam interaction, the matrix formulation of the FEA equations, and solution approximation schemes for dams excited by the seismic events. Two major time integration method classes in dynamic analyses: explicit and implicit methods are covered and illustrated with algorithm steps.

We have primarily focused on topics that affect numerical stability and accuracy. These include: the effect of time step on numerical stability, sensitivity of results to mesh fineness and element-level numerical integration schemes, and degradation of accuracy due to element type and mesh distortion. These topics span a wide range from physical and mathematical modeling, passing through FE model discretization, and getting to numerical results.

15. SUBJECT TERMS :

16. SECURITY CLASSIFICATION OF: UU			**17. LIMITATION OF ABSTRACT:** SAR	**18. NUMBER OF PAGES** 64 (not including Appendix)	**19a. NAME OF RESPONSIBLE PERSON:** Lan Nguyen
a. REPORT UU	**b. ABSTRACT** UU	**c. THIS PAGE** UU			**19b. TELEPHONE NUMBER** 303 445 2128

S Standard Form 298 (Rev. 8/98)
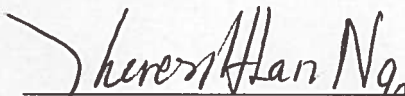P Prescribed by ANSI Std. 239-18

Bureau of Reclamation

Dam Safety Technology Development Program

Waterways & Concrete Dams Group 1

DSO-2018-09

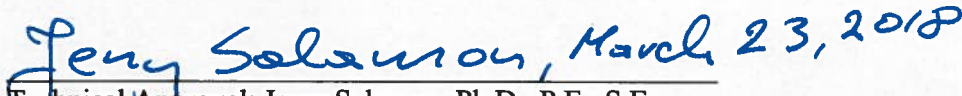# Guide for Analysis of Concrete Dam Structures using Finite Element Methods

)hereriHan Ng

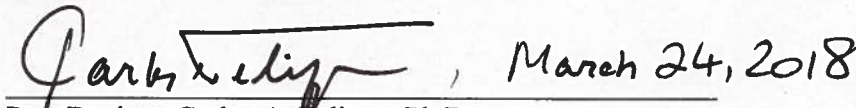Prepared Lan Nguyen, Ph.D./P.E.
Waterways & Concrete Dams Group 1, 86-68110

Jeny Saloman

Checked Jerzy Salamon, Ph.D., P.E., S.E.
Waterways & Concrete Dams Group 1, 86-68110

Jeny Salamon, March 23, 2018

Technical Approval: Jerzy Salamon, Ph.D., P.E., S.E
Waterways & Concrete Dams Group 1, 86-68110

Carly eliy , March 24, 2018

Peer Review: Carlos A. Felippa, Ph.D.
Professor of Aerospace Engineering
College of Engineering and Applied Science
University of Colorado Boulder, CO

CONTENTS

Tables

Figures

Acronyms and Abbreviations

FE	finite element
FEA	finite element analysis
FEM	finite element method
lbf	Pound Force Pascal
MPa	Mega Pascal
Reclamation	Bureau of Reclamation
TSC	Technical Service Center
USBR	Bureau of Reclamation

Keywords

Finite element analysis, Finite element methods, concrete dam structural analysis, concrete dam finite element models

Team Members, Partners, Reviewers

Name	Role	Mail Code & Contact Information
Lan Nguyen	Principal Investigator	Reclamation, TSC 86-68110, 303-445-2128 LTnguyen@usbr.gov
Jerzy Salamon	Technical Advisor	Reclamation, TSC 86-68110, 303-445-3219 JSalamon@usbr.gov
Carlos Felippa	Peer Reviewer (External Reviewer)	University of Colorado Boulder

I. Introduction

A.   Background

Finite element (FE) analysis has become a primary tool in structural evaluation of concrete dams. Expanded capabilities of the FE software allow modeling failures of concrete dams for static and seismic loads and to simulate the potential failure modes (PFM) of the structures. Results of such analyses are used as an input for risk estimations of the potential failure probability in the dam safety investigations. The approach of using non-linear FE analysis is adopted by the Bureau of Reclamation (Reclamation) as a Best Practice in the structural evaluation and risk assessment for concrete dams.

B. Specific problems

The FE approach for structural analyzes of concrete dams is embedded in a general procedure that includes the mathematical formulation of the problem, the FE discretization of the mathematical formulation, and the effective numerical solution of the problem using computers.

An analysis of concrete dams, particularly during earthquakes, require implementations of advanced mathematical models expressed by a system of partial differential equations and employment of complex numerical procedures for finding a solution of the problem (which usually is found to be nonlinear problems for modeling behavior of concrete dams). The advance analysis techniques differ significantly from the techniques used in the common engineering practice primary. Some specific aspects are listed as following:

Road map for analyzing concrete dams

An advanced structural analysis of concrete dams requires. The general scheme of the procedure is presented in a form of a flowchart in Figure 1.
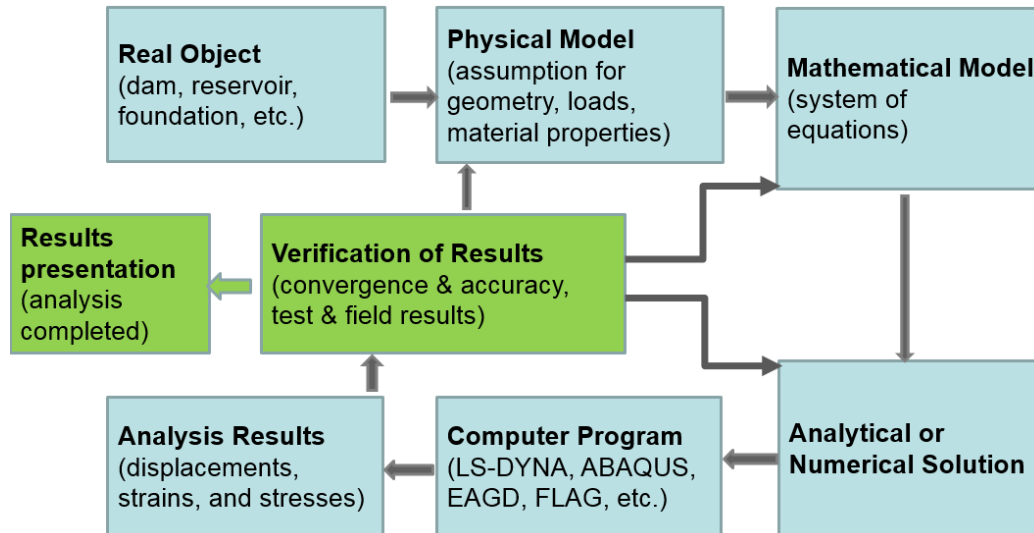
Figure 1. Flow chart of non-linear analysis for concrete dams

Accuracy of the analysis

It is important to understand the analysis process in any advanced analysis of concrete dams. Figure 1 presents a flow chart of the non-linear analysis process. As seen, the outcome of the analysis process is only the approximate solution. Therefore, the evaluation of the accuracy of the solution and the error estimation is essential.

Stress and strain measures

In the linear structural analysis, strains and stresses are defined by measures called *infinitesimal strains* and *Cauchy stress*, respectively. Informal terms are "engineering strains" and "engineering stresses". When moving to nonlinear analysis, those measure are replaced by "finite strains" and "energy-conjugate stresses", respectively. All of these measures form tensors. There is no unique finite strain measure for nonlinear analysis, but two commonly used in practice are Green strains and Almansi strains. Once the finite strain measure is chosen, the corresponding energy-conjugate stress is uniquely defined. For the Green strain measure, which is often used in FEM, the corresponding stress is called "second Piola-Kirchoff stress" often abbreviated to PK2.

Table 1 defines the strain measures and illustrates variances using different strain definitions for an example of a rod been in tension.

Table 1. Computed value of engineering, true, Green, and Almansi strains for a steel rod specimen in tension

Ratio ℓ / ℓ_o	Engineering strain $\Delta\ell / \ell_o$	Natural (true) strain $\ln(\ell / \ell_o)$	Green strain $\frac{1}{2}\frac{\ell^2 - \ell_o^2}{\ell_o^2}$	Almansi strain $\frac{1}{2}\frac{\ell^2 - \ell_o^2}{\ell^2}$
0.1	-0.9	-2.3	-0.495	-49.5
0.5	-0.5	-0.69	-0.375	-1.5
0.9	-0.1	-0.105	-0.0995	-0.11728
0.99	-0.01	-0.01005	-0.00995	-0.01015
1.01	0.01	0.00995	0.01005	0.00985
1.1	0.1	0.095	0.105	0.087
2.0	1.0	0.69	1.5	0.375
10.0	9.0	2.3	49.5	0.495

As seen, Table 1 shows that the differences in strain measures raise as the stretch ratio of the rod increases. This simple test demonstrates an importance of using a proper terminology in geometric non-linear (large deformation) formulation when performing analysis of concrete dams. In addition, it shows the significance of proper engineering interpretations of the analysis results.

C. Purpose

The overall objective of this report is to present general aspects of the FEM and provide a basis for the understanding of the solution process when it is applied to the advanced analysis of concrete dams. It is important to state here that this report serves as a tutorial to better understand the fundamental theory concept behind a commercial FEM codes. While the FE codes (ABAQUS, ANSYS, DIANA, LSDYNA, etc.) are specially intended for the material nonlinear analysis, this tutorial is restricted to linear and isotropic material models, its examples are to concrete gravity dams. A study on non-linear material models in the FE analysis of concrete dams could be found in the Reclamation's document Evaluations of Nonlinear Material Models of Concrete Dam using Finite Element Analysis [1].

In brief, the main goal of this study and referenced study [1] are to develop specific guidelines that help the analysts applying the FEM in the advanced structural analysis and then use the analysis results in the Dam Safety evaluation of Reclamation's concrete dam structures. In order to address the above goals, the following items topics tasks are formulated:

1. Formulate mathematical models describing behavior of a dam-foundation-reservoir system for seismic loads.

2. Discuss numerical methods for solving the mathematical models using the finite element.

3. Discuss the accuracy of the numerical solution.

4. Illustrate using examples the critical items that control the accuracy of the FE solutions.

5. Outline an analysis approach currently used by the Technical Service Center (TSC) in developing FE models of concrete dams and discuss the alternative approaches.

6. Develop a template to generate a FE input file of a gravity dam model to gain efficiency in building FE models. As such, the task is broken down in to two parts:

   - A template using TrueGrid mesh generator.

   - A FE modeling of a concrete dam configuration using LS-Prepost.

D. List of primary references used in the study

Work for this report utilizes the following references and previous studies:

1. The general guidelines for performing non-linear FE analysis for concrete dams provided in Reclamation's document entitled "State-of-Practice for the Nonlinear Analysis of Concrete Dams." [2] This document defines general information for non-linear structural analyses of concrete dams for seismic loads, however, no specific procedures are given on how to perform the analysis, validate the models, and evaluate and interpret the analysis results.

2. The 1982 "Finite Element Analysis Procedures in Engineering Analysis" [4] published by K-J. Bathe is a comprehensive guide for the numerical methods referenced frequently in this report. The referenced publications helps explain the mathematical principals and provides practical tips for implementing the FE method in analysis of concrete dam.

3. Reclamation's DSO-2014-08 report, "Evaluation of Non-linear Material Models in Concrete Dam FEA", by Jerzy Salamon and David Harris, investigates the non-linear behavior of concrete material for dams and verifies the parameters used in damage function of concrete material models in LS-DYNA's [1].

4. United States Society of Dams (USSD) paper "Numerical Assessment of Hydrodynamic Loads Induced during Seismic Interaction between Reservoir and Concrete Dams" by Jerzy Salamon layouts a concept of the FE formulation for a dam/reservoir interaction problem using "acoustic fluid" and "fluid-like material model" approaches [5].

II. Physical and mathematical models of a dam-reservoir system

A. Physical Model

The first step of any structural evaluation is to define a physical model (See Figure 1) based on a real structure (object). In general, the physical model can be described as a virtual model of the real structure that represents its behavior. . All characteristics of the real structure including the geometry of the object, material properties, and loads, are idealized in the physical model by dimensions, mechanical and thermal properties of the material, with static or time dependent loads.

B. Mathematical Model

For a given physical model of the dam-foundation-reservoir system, a mathematical model in the form of partial differential equations in space and time is formulated. The mathematical model represents the equilibrium requirements, constitutive relations, and continuity requirements, supplemented by the boundary conditions and, in dynamic analysis, initial conditions for the system.

The mathematical problem could be classified in a form of differential equations as elliptic, parabolic, or hyperbolic. Each of the class of equations is associated with different physical problem and the character of the solutions is distinctly different for the three categories of equations. In general, the elliptic differential equations govern the steady-state response of the system called also boundary value problems. In the parabolic and hyperbolic equations time, as an independent variable, is included and thus define propagation problems called initial value problems.

Using the formulation presented in [5] for the physical model shown in Figure 2, the system of equations for a dam-foundation-reservoir system during earthquakes could be expressed in the Eulerian description by equations of motions (Eq.1), geometric equations (Eq.2), material constitutive law for the dam and foundation (Eq.3), constitutive law for fluid (Eq.4), together with boundary conditions (Eq.5) and initial conditions (Eq.6).
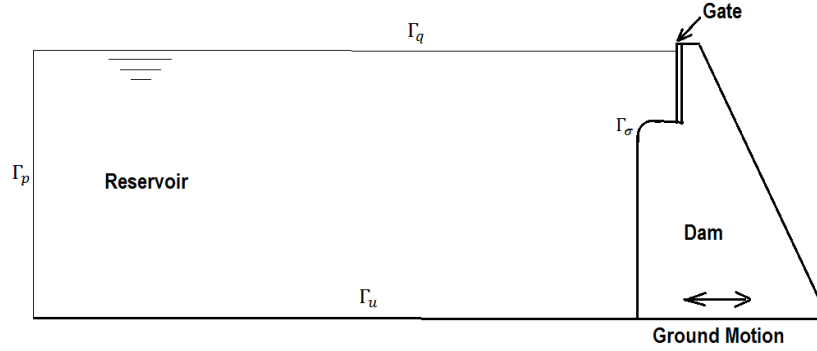
Figure 2. Model of a Dam - Reservoir System

$$\sigma_{ij,j} + \rho b_i = \rho \ddot{u}_i \qquad \text{where } i,j,k = 1,2,3 \qquad \text{(Eq.1)}$$

$$\varepsilon_{ij} = \frac{1}{2}\left(u_{i,j} + u_{j,i} - u_{k,i}u_{k,j}\right) \qquad \text{(Eq.2)}$$

$$\sigma_{ij} = \frac{E}{1+v}\left(\varepsilon_{ij} + \frac{v}{1-2v}\varepsilon_{kk}\delta_{ij}\right) \qquad \text{(Eq.3)}$$

where: $\underline{u}$, $\underline{\varepsilon}$, $\underline{\sigma}$ are displacement, Almansi finite strain tensor, Cauchy stress tensor. Cauchy stress could also be expressed by the second Piola-Kirchhoff stress tensor $S_{ij} = J^{-1}F_{ik}\sigma_{kl}F_{jl}$. Here, F_{ij} is the deformation gradient tensor, J is its determinant.

b_i are body forces

E, v, and ρ are elastic modulus, Poisson's ratio, and mass density, respectively, of concrete or foundation material.

Indices i, j, k, l range from 1 to 3 and the summation convention is assumed.

Behavior of the fluid is described in this report by a "fluid-like" material model (implemented in LS-DYNA software and currently used by Reclamation for analyzing the reservoir-concrete dam interaction behavior). In this formulation, fluid is defined as a structural elastic material (Eq.4) with the shear modulus equal to zero.

$$\dot{p} = \tfrac{1}{3}\dot{\sigma}_{kk} = K\,\dot{\varepsilon}_{kk} \tag{Eq.4}$$

where: $\dot{\varepsilon}$, $\dot{\sigma}$, and $\dot{p}$ are strain, stress, and pressure rates

K is the bulk modulus of fluid

$$\sigma_{ij}n_j = T_i \qquad\qquad on\ \Gamma_\sigma \tag{Eq.5}$$
$$\bar{q} = 0 \qquad\qquad on\ \Gamma_q$$
$$u_i = 0 \qquad\qquad on\ \Gamma_u$$
$$p = \bar{p} \qquad\qquad on\ \Gamma_p$$

where: Γ_σ, Γ_q, Γ_u, Γ_p are parts of the boundary shown in Figure 2.

Initial Conditions.

$$u_i(x, t_0) = u_i^0(x) \tag{Eq.6}$$
$$_i(x, t_0) = v_i^0(x)$$
$$\ddot{u}_i(x, t_0) = a_i^0(x)$$

Contact Conditions with Friction

The unilateral contact is described by a contact distance function:

$$d_n = \left(x_p - x\right)\bar{n} \tag{Eq.7}$$

where x_p is the projection of x on the deformed potential contact surface.

The unilateral contact low has the form

$$d_n > 0 \quad then \quad t_n = 0 \ ; \quad d_n = 0 \quad then \quad t_n < 0 \tag{Eq.8}$$

$$d_n \cdot t_n = 0 \qquad\qquad ; \quad t_n < \sigma_{ij}n_in_j$$

For the classical formulation of the friction condition

$$t_T - \mu \cdot t_n \le 0 \ on\ \Gamma_\sigma \tag{Eq.9}$$

$$if \qquad t_T - \mu \cdot t_n < 0 \quad then \quad \dot{u}_T^w = 0$$

$$and \qquad t_T - \mu \cdot t_n = 0 \quad then \quad \dot{u}_T^w = -\lambda \cdot t_T$$

where $\dot{u}_T^w$ is difference in sliding velocity at the contact surface, $t_T = \sigma_{ij}n_j - t_n n_i$, and μ - is the friction coefficient.

In the 3-dimensional Cartesians space, the displacement vector u consists of 3 components. The 3x3 stress σ and the strain ε tensors contain 9 components each but only 6 components are independent because of the symmetry of these tensors.

The mathematical problem (Eq. 1 through Eq. 3) is defined by 9 partial differential equations (3 equations of motions and 6 independent equations describe a relation between displacements and strains) together with 6 constitutive equations providing relations between stresses and strains. Total 15 unknowns are determined by 15 equations.

The mathematical formulation given by equations 1 through 9 could also be defined in the Lagrangian Description and expressed using 2nd Piola-Kirchhoff stress and Green strain tensors.

For clarification purposes, the seismic load is introduced in (Eq. 1) by the component $\rho \ddot{u}_t$. In general, three types of nonlinearities are incorporated in the system of equations, this includes geometric nonlinearities (Eq. 2), physical nonlinearities (Eq.3), and contact with friction conditions (Eq. 7 through Eq. 9). More information on non-linear material models for concrete dams could be fined in Reclamation's technical memorandum prepared by Salamon and Harris referenced in [1].

C. Variational Formulation

The system of governing equilibrium equations presented in Section B could be alternatively expressed by in a form of the virtual work formula. The variational approach is very effective technique when a numerical analysis is employed. In the formulation, total potential Π(u) of the system is defined. The solution is obtained by finding the vector of displacements **u** corresponding to the potential at its extremes, maintained by the condition when the change in potential energy with respect to the change of displacement δΠ(δu) = 0. For a specific linear case, where potential is defined as a difference between the strain energy of the system and the total work of load, $\Pi = \mathcal{U} - \mathcal{W}$, the solution for displacements **u** corresponds to the minimum of Π(u). Applying the procedure for derivation of the virtual power principle, the variational formulation for equilibrium equations takes the form:

$$\int_\Omega \sigma_{ij}(u)\,\varepsilon_{ij}(v)\,dV + \int_\Omega \rho\,\ddot{u}_i v_i\,dV$$

$$= \int_\Omega \rho\,b_i v_i\,dV + \int_{\Gamma_c} p_i\,v_i\,d\Gamma + \int_{\Gamma_\sigma} t_n\,v_n^w\,d\Gamma$$

$$+ \int_{\Gamma_c} t_T\,v_T^w\,d\Gamma \qquad\qquad \forall(v) \in V$$

(Eq.10)

where $v_n^w = \dot{u}_n^w$ and $v_T^w = \dot{u}_T^w$ denote the normal and tangential components of the velocity vector.

D. Incremental Formulation of Equation of Motions

In the non-linear analysis, the equilibrium of the body is considered at the discrete time points *0, Δt, 2Δt, ..., t,* and the equilibrium is postulated in the "current" configuration *t+Δt.* The equations of motions are expressed in the time domain by time increments *Δt.* Displacements, strain, and stress tensors at time *t+Δt* are expressed as:

$$u^{t+\Delta t} = u^t + \Delta u \qquad for\ diplacements$$
$$\varepsilon_{ij}^{t+\Delta t} = \varepsilon_{ij}^t + \Delta\varepsilon \qquad for\ strains$$
$$\sigma_{ij}^{t+\Delta t} = \sigma_{ij}^t + \Delta\sigma \qquad for\ stresses$$

(Eq.11)

$\Delta\boldsymbol{u}$, $\Delta\varepsilon$, and $\Delta\boldsymbol{\sigma}$ denote increments of displacements, strains, and stresses, respectively, corresponding to the time increment Δt.

III. Solution of the Mathematical Problem

A. Finite Element Formulation

Applying the standard procedure for derivation of the virtual power principle and considering geometric relationships between the displacements and strains, constitutive equations, and the initial conditions, the finite element discretization of the Galerkin variational formulation of the preceding equations results in the system of second order ordinary differential equations (Bathe 1982) with constant coefficients for time t in the form:

$$M \, \Delta\ddot{u}(t) + C\Delta\dot{u}(t) + K\Delta u(t) = \Delta R(t) \qquad (\text{Eq.12})$$

where: M, C, and K, are the mass, damping, tangent stiffness matrices, respectively

The unknown vector of nodal displacements u(t) represents the relative displacements at the nodes of the FEelement. Matrix M has constant elements. The forcing vector R(t) contains external forces including the ground acceleration.

B. Analytical Solution

Analytical solutions, also called "closed form solutions", are more intellectually satisfying, particularly if they apply to a wide class of problems. Unfortunately, they tend to be restricted to regular geometries and simple boundary conditions. Additionally, either most problems faced by engineers do not yield to exact analytical solutions, or to achieve such exact analytical solutions, it would require a disproportionate amount of effort. The (more) practical way of seeking for solution of the mathematical problem is through numerical simulations. This is where FEM take place [6].

C. Numerical Solution (Discretization)

In the numerical solution of a mathematical problem, FE reduces the number degree of freedoms of the system to a finite number. This reduction is essentially a discretization. Discretization can proceed in time dimensions as well as in space dimension.

1. Discretization in time:

The mathematical problems could be subdivided according to whether inertial effects are taken in to account or not. According to [6], these problems are generalized as continuum mechanics problems, and are categorized into the followings Table 2.

Table 2. Subdividing Static and Dynamic problems

Continuum mechanics	Statics	Time invariant
		Quasi-static
	Dynamics	

In statics, inertial forces are neglected. Static problem can be sub classified into time invariant and quasi statics.

In dynamics, the time dependence is strictly considered because the calculation of inertial forces requires derivatives respect to actual time to be taken. Dynamic analysis is needed when the variation of displacement with time is so rapid that inertial effects cannot be ignored. Practical examples for these analyses are: earthquakes, vehicle crashes, rocket launches, etc [6].

There are two common approaches used in solving Eq. 12 with respect to time, namely: explicit and implicit approximations. In the continues time domain discrete time points are *0, Δt, 2Δt, ..., t,* identified at which the equilibrium of the body is assumed and the equilibrium is postulated in the "current" configuration *t+Δt*. Displacements are expressed as a function of as:

$$u\ (t + \Delta t) = u(t) + \Delta u \qquad \text{(Eq.13)}$$

$\Delta \boldsymbol{u}$ denote increments of displacements corresponding to time increment Δt.

2. *Explicit Approximation*

2.1 Forward Euler method
Given $u(t)$ is a displacement function with respect to discrete point of time t. The displacement at the time $(t\ + \Delta t)$ in the Forward Euler method is expressed as:

$$u\ (t + \Delta t) = u(t) + \frac{du(t)}{dt}\Delta t = u(t) + f(u, t) * \Delta t \qquad \text{(Eq.14)}$$

This method uses the slope function $f(u, t)$ at each point of time, extrapolates, and finds the result for the displacement in the next discrete point of time. With a given slope $f(u, t)$ of the displacement function and the time increment Δt, one can explicitly calculate the displacement at the next time step $(t + \Delta t)$. Therefore, this method is called *explicit method* (Bathe 1982 [4]).

2.2 Central Difference Method
The other type of the explicit method is the central difference method (CDM). A general scheme used in CDM is illustrated in the following figure.
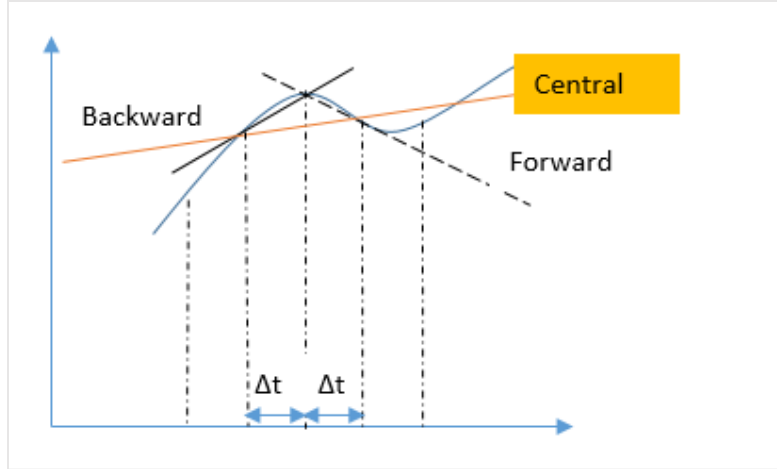
Figure 3. Interpretation of CDM scheme

Using the CDM approach, the change in displacement with respect to time can be expressed as:

$$\frac{du}{\partial t} = \left\{ \frac{-u^{t-\Delta t} + u^{t+\Delta t}}{2\Delta t} \right\} \qquad \text{(Eq.15)}$$

Notice that the CDM span over two interval time steps as shown previously in **Figure 3**.

Apply Taylor expansion for the forward difference for the change of displacement with respect to time

$$\frac{du}{\partial t} = \frac{u^{t+\Delta t} - u^t}{\Delta t} - \frac{\Delta t}{2} * \frac{d^2 u^t}{dt^2} - \frac{\Delta t^2}{6} * \frac{d^3 u^t}{dt^3} + \cdots \qquad \text{(Eq.16)}$$

Apply Taylor expansion for the backward difference for the change of displacement with respect to time

$$\frac{\partial u}{\partial t} = \frac{u^t - u^{t-\Delta t}}{\Delta t} + \frac{\Delta t}{2} * \frac{\partial^2 u}{\partial t^2} - \frac{\Delta t^2}{6} * \frac{\partial^3 u}{\partial t^3} |i + \cdots \qquad \text{(Eq.17)}$$

The summation of the forward difference and backward difference shown in the two equations above is:

$$\frac{\partial u}{\partial t} = \frac{-u^{t-\Delta t} + u^{t+\Delta t}}{2\Delta t} - \frac{\Delta t^2}{6} * \frac{\partial^3 u}{\partial t^3} + \cdots \qquad \text{(Eq.18)}$$

Equation Eq. 18 yields what shown in Eq 15 previously, with the following truncated error:

$$Truncated\ error = E_{trunc}(f, \Delta t) = O(\Delta t^2) \qquad \text{(Eq.19)}$$

Acceleration is derived by the change of velocity $(\frac{\partial u}{\partial t})$ with time:

$$\frac{\partial^2 u}{\partial t^2} = \frac{\frac{u^{t+\Delta t}-u^t}{\Delta t}-\frac{u^t-u^{t-\Delta t}}{\Delta t}}{\Delta t} = \frac{u^{t+\Delta t}-2u^t+u^{t-\Delta t}}{\Delta t^2} \qquad \text{(Eq.20)}$$

Substitute Equation Eq. 22 and Eq. 23 into Equation Eq. 32, grouping the term with **M**, **C**, and **K**, we have:

$$\mathbf{M}\left(\frac{u^{t-\Delta t}-2u^t+u^{t+\Delta t}}{\Delta t^2}\right) + \mathbf{C}\left(\frac{-u^{t-\Delta t}+u^{t+\Delta t}}{2\Delta t}\right) + \mathbf{K}u^t = \mathbf{P}^t \qquad \text{(Eq.21)}$$

Regrouping the equation in the order of $u^{t+\Delta t}$, u^t, and $u^{t-\Delta t}$, we have:

$$\left(\frac{1}{\Delta t^2}\mathbf{M} + \frac{1}{2\Delta t}\mathbf{C}\right)u^{t+\Delta t}$$

$$= \mathbf{P}^t - \left(\mathbf{K} - \frac{2}{\Delta t^2}\mathbf{M}\right)u^t - \left(\frac{1}{\Delta t^2}\mathbf{M} - \frac{1}{2\Delta t}\mathbf{C}\right)u^{t-\Delta t} \qquad \text{(Eq.22)}$$

As seen above, the solution of $u^{t+\Delta t}$ is calculated from the solution of the equilibrium conditions at time *t* and the previous time *t-Δt*. This agrees with the definition on the *explicit method*.

3.   Implicit Approximation

The generic form for the implicit method is as follow:

$$u\,(t + \Delta t) = u(t) + f\,(u(t + \Delta t), t + \Delta t) * \Delta t \qquad \text{(Eq.23)}$$

In the implicit approach, displacements at the current time $(t + \Delta t)$ are expressed by the displacements at the time t and the displacement increments Δt.

Two numerical solution schemes using iterative process are presented in this section, a full Newton Method and a modified Newton Method.

3.1   Full Newton method

The Full Newton (also called Newton-Raphson (NR) or Newton method) is a powerful technique for solving system of nonlinear equations. Newton gave a version of the iterative method in 1669. Raphson generalized and presented the method in 1690. Both mathematicians used the same concepts, and both algorithms gave the same results [7].

For illustration of the NR solution method, we consider an example of a nonlinear spring that is subjected to an external force F^{ext}, and internal force F^{int}, in solving for the unknown displacement "u", the following equilibrium must be satisfied:

$$F^{int}(u) = F^{ext} \qquad \text{(Eq.24)}$$

Suppose we have the internal force defined by a non-linear function of displacement:

$$F^{int}(u) = au^3 + bu^2 + cu \qquad \text{(Eq.25)}$$

From the equilibrium equation, define a force residual r as a function of the displacement u,

$$r(u) = F^{ext} - F^{int}(u) \qquad \text{(Eq.26)}$$

There exists an exact solution u^* such that the force residual equilibrium $r(u^*)=0$. The solution is approached by an **iterative** method, while the previous iterate is defined as u^k, and the current guess is $u^k + \delta u^k$ where $u^* = u^k + \delta u^k = u^{k+1}$

Apply Taylor series expansion about the iterative u^k, we have:

$$r(u^k) + \frac{\partial r(u^k)}{\partial u}\delta u^k + \frac{\partial^2 r(u^k)}{2\partial u^2}(\delta u^k) \qquad \text{(Eq.27)}$$

$$+Higher\ order\ truncated\ error = 0$$

3.1.1   Consistent tangent

If we drop the quadratic term in and the higher order truncated error (Eq.39), our linearization become a set of $r(u^k) + \frac{\partial r(u^k)}{\partial u}\delta u^k = 0$. In numerical method notation, the $\frac{\partial r(u^k)}{\partial u}\delta u^k$ term is called *consistent tangent* at the iteration k.

At the iteration k, the solution is updated as:

$$r(u^k) = F^{ext} - F^{int}(u^k) \qquad \text{(Eq.28)}$$

And, the change in the displacement at the iteration k is:

$$\delta u^k = [r(u^k)] * \left(\frac{\partial r(u^k)}{\partial u}\right)^{-1} \qquad \text{(Eq.29)}$$

From the residual equation, the change of the residual equation with respect to change of displacement $\frac{\partial r(u^k)}{\partial u}$ is the same as $\frac{\partial F^{int}(u^k)}{\partial u}$; Therefore,

$$\delta u^k = (\frac{\partial F^{int}(u^k)}{du})^{-1}[r(u^k)] \qquad \text{(Eq.30)}$$

3.1.2   Convergence check

At end of every guesses, there is a validation check which indicates if the "guess" value is approaching the true solution. This step is called a *convergence check*.

IF $\frac{|r(u^{k+1})|}{|r(u^0)|} <$ tolerance THEN $u^* = u^{k+1}$

ELSE continue iteration

Figure 4 presents an illustration for an incremental solution that FEA uses in a "guessing process" for results of a non-linear function.
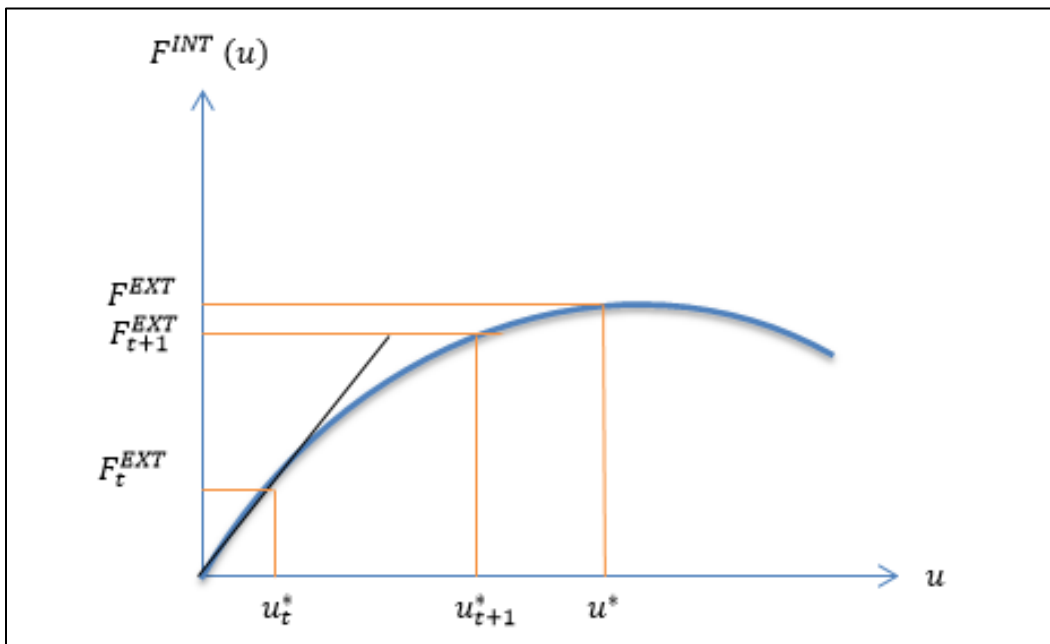
Figure 4. Incremental solutions solved by FEA for a non-linear function

From the example, it is seen that the convergence criteria measures how well the obtained solution satisfies equilibrium. Often, we use: (1) Energy criteria, (2) Force criteria, or (3) Displacement criteria [2].

On energy criteria, ETOL (a constant energy tolerance value, [6] defines $R^{t+\Delta t} - F^{t+\Delta t}$ as a generalized force and also defines a inner product of $(R^{t+\Delta t} - F^{t+\Delta t})^T u$, meaning the external energy or work) is set in advanced such that the solution satisfies:

$$\frac{\Delta u(R^{t+\Delta t} - F^{t+\Delta t})}{\Delta u(R^{t+\Delta t} - F^t)} \leq ETOL \qquad (Eq.31)$$

On force or moment criteria, RTOL (a constant reaction tolerance) is set in advanced such that the solution satisfies:

$$\frac{\|R^{t+\Delta t} - F^{t+\Delta t}\|}{RNORM} \leq RTOL \qquad (Eq.32)$$

On displacement criteria, DTOL (a constant displacement tolerance) is set in advance such that the solution found in the analysis satisfies:

$$\frac{\|\Delta \bar{u}^{(t)}\|_2}{DNORM} \leq DTOL \qquad (Eq.33)$$

Evidently, tolerance value must follow certain conditions and is defined as an acceptable range. The following figure shows a snap shot of the LS-DYNA input card requesting solution output with the displacement, force, and energy tolerances are set arbitrary at 0.01.

```
*CONTROL_IMPLICIT_SOLUTION
$#   nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
          2         5        30  0.010000  0.010000     0.000     0.000     0.000
$#    dnorm     diverg     istif   nlprint    nlnorm   d3itctl     cpchk
          2         1         1         0         2         0         0
$#   arcctl     arcdir    arclen    arcmth    arcdmp
          0         0     0.000         1         2
```

Figure 5. Figure shows the displacement tolerance (dctol) and energy tolerance (ectol) values are set arbitrary at 0.01 in the control solution card used in LS-DYNA FEA

3.2.1   Modified Newton method (need to specify in LS-DYNA options)

The residual equation for the Modified Newton Raphson (MNR) method is written as:

$$r(u^k) = F^{ext} - F^{int}(u^k) \qquad (Eq.34)$$

The consistent tangent is as follow:

$$\delta u^k = \left(\frac{\partial F^{int}(u^0)}{\partial u}\right)^{-1} [r(u^k)] \qquad (Eq.35)$$

The following figure demonstrates the iterations in solving a non-linear equation with the MNR method. Notice the constant consistent tangent slopes.
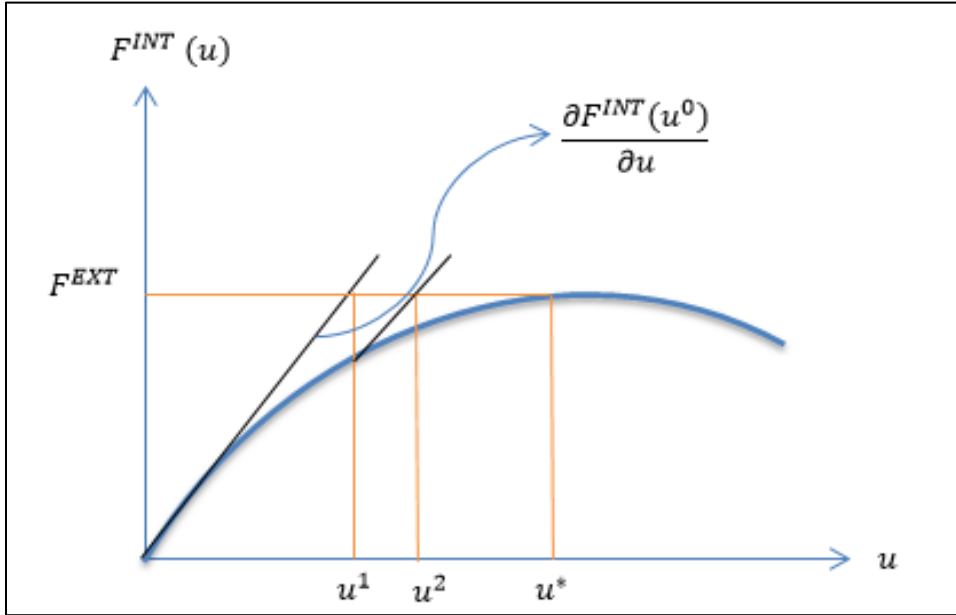
Figure 6. Modified Newton Raphson iteration method

A powerful modification to the Newton Raphson method is a class of methods called "arc-length" methods. According to [8], this is the most typical example of a nonlinear solution scheme which considers simultaneous iteration on both the load and displacement variables. The premise of the method is to constrain the solution path to an arc length, Δs_j^i. The important key of the method is introducing a dimensionless load control parameter, $\Delta \lambda_j^i$. The arc length is calculated via a norm of the increment $(\Delta u_j^i, \Delta \lambda_j^i)$. The general constraint in determining the arc length is: $\Delta u_j^i . \Delta u_j^i + n(\Delta \lambda_j^i)^2 = (\Delta \lambda_j^i)^2$, with $\in [0,1]$.

Figure 7 and Figure 8 below show the nonlinear functions $F(u)$ that is solved by two methods: the NR method and the Modified NR method with using Load Control scheme. In general, the two methods give good results as compared to the exact solution. However, when encounters the negative slope, the consistent tangent K matrix used in the NR scheme tends to jump apart and can only search for solutions if the slope remains positive. In the Modified NR method, the consistent tangent K is kept constantly through the entire approximation process and thus approximates the displacements better even when the loading encounters with negative slope. It is important to note here that, the MNR method with load control scheme can solve for the solution particular in this single degree of freedom (DOF) example. In real dam structure models, where the model could have more than 10,000 DOFs, the arc length method is highly recommended.   For demonstration and algorithm, readers are referred to [8].

Figure 7 shows the "jump" resulting from the inability of solving for displacement when approaching the negative slope in the original NR solving scheme.

Nonlinear F function

Notice the "difficulties" (resulting in no solutions) in convergence when the consistent tangent K approaches the negative slope
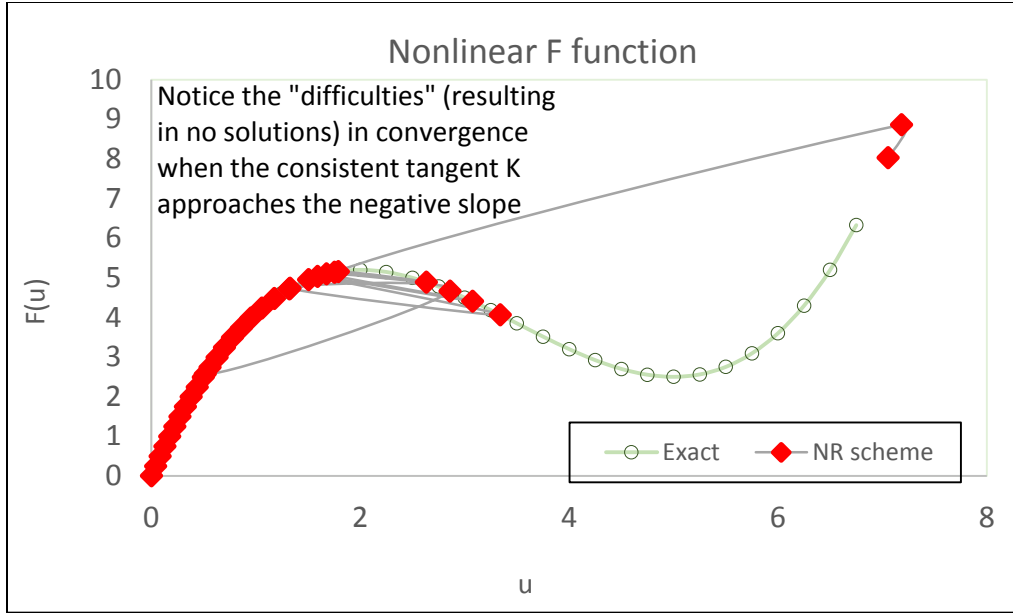
Figure 7. Example of using Newton Raphson method in solving for a nonlinear function

To overcome this issue, the modified Newton Raphson method is used (could be specified with LS-DYNA options). As seen in the following Figure 8 where the positive tangent stiffness is "repeated" and displacement is updated throughout the approximation process.

Nonlinear F function
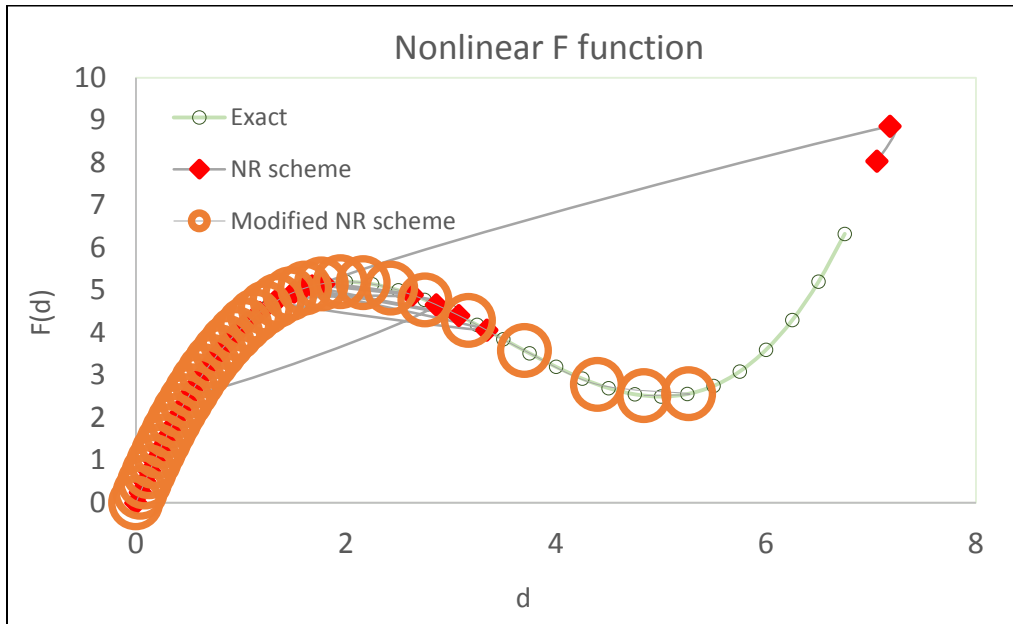
Figure 8. Example of using Modified Newton Raphson method with load control scheme to solve for a nonlinear function for a SDOF example

3.2.2   *When to use the Newton Raphson or arc length in the analysis?*

When perform seismic analysis, it is important to notice that a loading process in dynamic dam analysis contains two stages: Stage 1 with static loads (e.g. gravity) and stage 2 with dynamic load. During stage 1, the static loading process is always at "loading" stage (constant). In search for structural responses under this load, the consistent stiffness tangent remains positive, the Newton Raphson method (also known as Linear Solver No. 1 option in Control_Implicit_Solution card in LS-DYNA) could be used.

During stage 2, when the loading and un-loading shakes from the dynamic motion in turns have both positive and negative slope. The Nonlinear Solver option No. 2 in Control_Implicit_Solution in LS-DYNA card (as shown in the following Figure 9) is recommended. For more detail definitions of solving methods, the reader is referred to the LS-DYNA Keywords Users' Manual [9].

```
*CONTROL_IMPLICIT_SOLUTION
$#   nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
          2         5        30  0.010000  0.010000     0.000     0.000     0.000
$#    dnorm     diverg     istif   nlprint    nlnorm   d3itctl     cpchk
          2         1         1         0         2         0         0
$#   arcctl     arcdir    arclen    arcmth    arcdmp
          0         0     0.000         1         2
```

Figure 9. Figure shows the Nonlinear Solver options (#nsolvr = 2) used in LS-DYNA

4. Discretization in Space

Discussed in the Introduction of Finite Element Method written by Felippa in [6], a final classification of computation of computational solid and structural mechanic convert a continuum mathematical model into a discrete model of finite number of degrees of freedom.

The following table sorts the uses of each spatial discretization method and their dominate areas

Table 3. Spatial discretization found in [4]

	Application areas	**Comments on the Efficiency and the age of method**
Finite Element Method	For linear and nonlinear problems	Currently dominates over all discretization methods
Boundary Element Method	For linear problems	
Finite Difference Method	For solid and structural mechanics problems	Virtually disappeared from practical use
	For fluid mechanics problems	Still important, but dominance has diminished over time
Finite Volume Method	Focus on the direct discretization of conservation laws, favored in highly nonlinear problems of fluid mechanics problems	Currently dominates over all discretization methods
Spectral Methods	Based on global transformations and Eigen composition of the governing equations, map the physical computation domain to transform spaces where problem can be efficiently solved	
Mesh free method	Finite different methods on arbitrary grids constructed using a subset of finite element techniques.	Recent new comer to the scene of spatial discretization

In this study, our focus is on the spatial discretization using finite element method. More background material related to FEM modeling, element attributes (dimensionality, nodes, geometry, etc.), and classification or mechanical elements are not discussed in this study. Selective references could however be found in [6], or in [7]. For nonlinear material aspects that involve consideration of using FEM or FVM, readers are referred to [1].

4.1   Discussion of integration points and element stiffness

In the FE discretization with respect to space, the unknown displacement vector is expressed by the shape function matrix $N(x)$ and the nodal displacement vector ΔU(t) as follow:

$$u(x,y)_{element} = N_i(x,y)u_i = N\ u^* \qquad \text{(Eq.36)}$$

Where:

$u(x,y)_{element}$ = displacement in the entire element

$N_i(x,y)$ = shape function

u_i = displacement of the i element node

i = index ranges over the element's nodes.

The shape functions $N_i(x,y)$ are linear, quadratic, or occasionally cubic polynomials that are chosen to be unity at node i and zero at the other element nodes. The shape function can be evaluated at any location within the element by the means of standard subroutines (node- to node), and so the approximation displacement of the entire element can be obtained.

Approximation for the strain and stress within an element are defined as:

$$\varepsilon = B_i u_i \qquad \text{(Eq.37)}$$

$$\sigma = DB_i u_i \qquad \text{(Eq.38)}$$

With D is a material coefficient matrix

$$B = \begin{bmatrix} N_{j,x} & 0 \\ 0 & N_{j,y} \\ N_{j,y} & N_{j,x} \end{bmatrix} \qquad \text{(Eq.39)}$$

The existence of displacement leads to the formulation of strain energy $\partial U = \int \partial \varepsilon^T \sigma\ dV$ where V is the volume of the entire element.

Put the stress and strain equations back to the equation for strain energy, we have:

$$\partial U = \int \partial (B_i u_i)^T DB_i u_i\ dV \qquad \text{(Eq.40)}$$

which could be re-formed to:

$$\partial U = \partial u_i{}^T \int B_i{}^T DB_i\ dVu_i \qquad \text{(Eq.41)}$$

By definition, an increase in strain energy must equal to the work done by a nodal force, that is:

$$\partial W = \partial u_i^T f_i = \partial U \qquad \text{(Eq.42)}$$

Simplify the incremental transposed displacement term, and combine the incremental strain energy and the definition for incremental work, one has:

$$\left[\int B_i^T DB_i \, dV \right] * u_i = f_i \qquad \text{(Eq.43)}$$

The $\left[\int B_i^T DB_i \, dV \right]$ term is also known as the element stiffness.

Finally, the integral $\int B_i^T DB_i \, dV$ must be determined within the volume of the element. Gauss-Legendre numerical integration is commonly used in FEA codes (also known as *Gauss point evaluation*). The integration consists of evaluating the integrand at optimally selected integration points within the elements, and forming a weighted summation of the integrand values at these points.

The integration point numbers are different according to each solid element type. In LSDYNA the following solid element types are defined:

- Triangular/tetrahedron 4 nodded element: 1to 4 integration points

- Pentahedron 6 nodded element: 1 to 2 integration points.

- Hexahedron/8 nodded element: 1 to 4 integration points, or fully integrated over selective surfaces.
- Hexahedron/27 nodded element: 1 to 4 integration points, or fully integrated over selective surfaces.

An illustration for the integration points in solid elements is shown as follow:
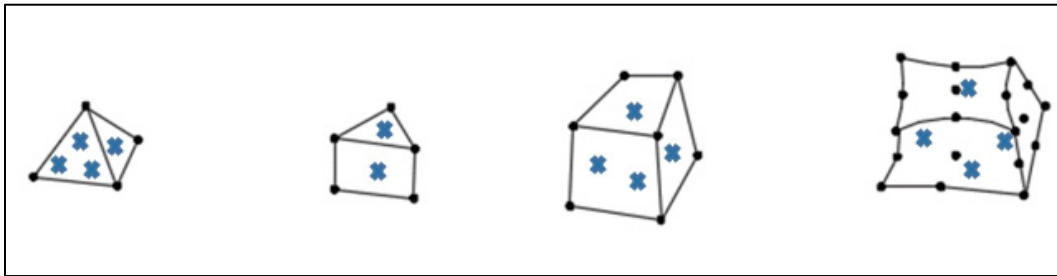
Figure 10. Solid elements and the integration points. (From left to right: tetrahedron 4 nodded element, pentahedron 6 nodded element, hexahedron 8 nodded element, and Hexahedron 27 nodded element)

The overall obtained displacements, stresses, or strains depend on the selection of the element types, size of the elements, and the actual number of the integration point(s). More detail discussion for the FEA theory particularly in this evaluation concept could be found in the 2001 FEA report prepared by David Roylance, MIT [10] or in any introduction of FE courses, referenced here is [11].

IV Accuracy Numerical Solutions

A. Accuracy for discretization in time

1. Accuracy, consistency, and stability conditions in numerical methods

In the numerical solutions of a mathematical problem in the time domain, it is important to maintain three conditions: (1) accuracy, (2) consistency, and (3) stability of the method.

1.1 Accuracy

Applying Taylor expansion on the explicit approximation equation:

$$x\left(t + \Delta t\right) = x(t) + f(x, t) * \Delta t + \frac{d^2 x(t)}{dt^2}\frac{\Delta t^2}{2!} + \frac{d^3 x(t)}{dt^3}\frac{\Delta t^3}{3!} + \cdots \qquad \text{(Eq.44)}$$

From the Taylor expansion result seen in Eq.48, the explicit method has local truncated error to the second order of its time step, i.e, L.T.E $= O(\Delta t^2)$. This means, in using the approximation method, one should use a "small" time step Δt such that in some senses, the $O(\Delta t^4) \leq O(\Delta t^3) \leq O(\Delta t^2)$.

1.2 Consistency

Consistency in approximation requires the limit of the truncated error reaches to zero when the time step approaches zero $(\lim_{\Delta t \to 0} \frac{d^2 u(t)}{dt^2}\frac{\Delta t^2}{2!} + \frac{d^3 u(t)}{dt^3}\frac{\Delta t^3}{3!} + \cdots) = 0$. With this, the explicit method (or Euler Forward in general) is consistent because the condition is satisfied for any $\Delta t > 0$ (See Eq. 48)

1.3 Stability

Stability requires the errors in the approximation analysis remain bounded. In the other words, it ensures that we are modeling a well-behaved physical system where there are no instability or no blow-up. Since the behavior of the numerical solutions depends on the behavior of the errors, the numerical solutions will blow up if the errors blow up. Consider the arbitrary equation:

$$\frac{d(u)}{dt} = -au \, , with \, u(t_0) = u_0 \qquad \text{(Eq.45)}$$

The analytical solution for the equation above is $u(t) = u_0 e^{-at}$. The explicit method solves for the solutions as follow:

$$u\,(t + \Delta t) = u(t) + \left(-a * u(t)\right) * \Delta t = (1 - a\Delta t) * u(t) \qquad \text{(Eq.46)}$$

So at the first time step Δt,

$$u\,(t_0 + \Delta t) = (1 - a\Delta t) * u_0 \qquad \text{(Eq.47)}$$

At the second time step,

$$u\,(t_0 + 2\Delta t) = (1 - a\Delta t)^2 * u_0 \qquad \text{(Eq.48)}$$

And, at the n time step,

$$u\,(t_0 + n\Delta t) = (1 - a\Delta t)^n * u_0 \qquad \text{(Eq.49)}$$

In this example, once the time step increases, the solutions of the equation should decay and get smaller (as a stays constant and the initial displacement u_0 being reduced) In order to stay in the "expecting-result", the bound for the magnification is conditionally set as $|1 - a\Delta t| < 1$. Solving for this condition for time step Δt, Euler Forward provides stability if the $\Delta t < \frac{2}{a}$, and $1 - a\Delta t < 1$ for any $\Delta t > 0$. *Because of the need for a condition, the Euler Forward is also known and referred as a **"conditional stable"** method.*

By contrast, the stability of the implicit method for the previous example is as follow:

$$\frac{d(u)}{dt} = -au \,, with\; u(t_0) = u_0 \qquad \text{(Eq.50)}$$

Applying Euler Backward to (Eq.54),

$$u\,(t + \Delta t) = u(t) + \left(-a * u(t + \Delta t)\right) * \Delta t \qquad \text{(Eq.51)}$$

$$u\,(t + \Delta t) = \frac{u(t)}{(1 + a * \Delta t)} \qquad \text{(Eq.52)}$$

Knowing that the solution is decaying, the factor $\frac{1}{(1+a*\Delta t)} < 1$ is true for any $\Delta t > 0$. The $\frac{d(u)}{dt} = -au$ example is an arbitrary one; however, from this example the implicit method is proven to be an ***unconditionally stable***.

2. *Illustration Accuracy Estimations for Discretization in Time- Response of a SDOF system*

In this section, we discuss accuracy of a solution for a single degree of freedom (SDOF) example subjected to a constant ramped up force. The comparison is between (1) the exact solution, (2) the implicit approximation (including the Newmark β method and the Wilson θ method), and (3) the explicit approximation (the Central Difference method). Table 4 lists six algorithms used to compute solutions for the SDOF example.

It is important to note that the accuracy investigation listed in Section IV A.1 refers to the truncation accuracy of the time integrator for a scalar SDOF equation. Accuracy in the space dimensions or for a system of equations is a far more involved subject.

Table 4. Algorithms and Numerical Methods Used in Computational Structural Dynamic Problems

Type	Algorithms		Numerical method used
Exact solution	Duhamel's Integral		Formulates exact solution
	Precise Exact Method		Formulates exact solution
Approximation	Implicit approximation	Newmark β method	Formulates approximation using numerical integration
	Implicit approximation	Wilson θ method	Formulates approximation using numerical integration
	Explicit approximation	Central Difference method (*)	Formulates approximation using numerical differentiation
	Explicit approximation	4th order Runge-Kutta Method	Formulates approximation using numerical integration

Table 5. Time step for stability criteria in explicit integration methods

Explicit Numerical Integration Method	Stability criteria	Restraint time step (critical value) (For SDOF) for stability
Central Difference Method	Conditionally Stable	$\frac{1}{\pi}T_0$
4th order Runge-Kutta Method	Conditionally Stable	No restraint for time step

As it is seen in Table 6, the time step for stability criteria in implicit integration methods, the explicit approximation for the SDOF requires the time step is smaller than a critical time step (T_{cr}).

Table 6. Time step for stability criteria in implicit integration methods

Implicit Numerical Integration Method	Stability criteria	Restraint time step (critical value) (For SDOF) for stability
Newmark β method	Conditionally Stable	$\frac{1}{\pi\sqrt{2}} * \frac{1}{\sqrt{\gamma - 2\beta}}T_0$
Wilson θ method	Unconditionally Stable	No restrain for time step, $\theta \geq 1.366$

For clarification, Newmark methods maybe conditionally stable or unconditionally stable in linear dynamics, depending on parametric choices ($\gamma\ or\ \beta$) as seen in Table 6. They are conditionally stable in nonlinear dynamics. In addition, Wilsons's method is unconditionally stable in linear dynamics, but may become locally unstable in nonlinear dynamics problems that include strong shocks.

The example is carried out for the SDOF and the displacements estimated via different approximation methods. Table 7 lists the information used in constructing of the model for the comparison between methods.

Table 7. Information used in SDOF example

Item	Value	Unit
Mass	104.3	Kip-s^2/ft
Spring stiffness	595,956.4	Kip/ft
External force	1000	Kips
Damping	0	
Time step (dt)	varies	seconds

Figure 11 shows damping free responses for a SDOF example. Here, a significantly smaller time step was used. As seen, with small time step (Δt=0.001 seconds), the explicit (CDM) and implicit (Newmark Beta and Wilson Theta) approximations provide solutions that match very well (Figure 11).
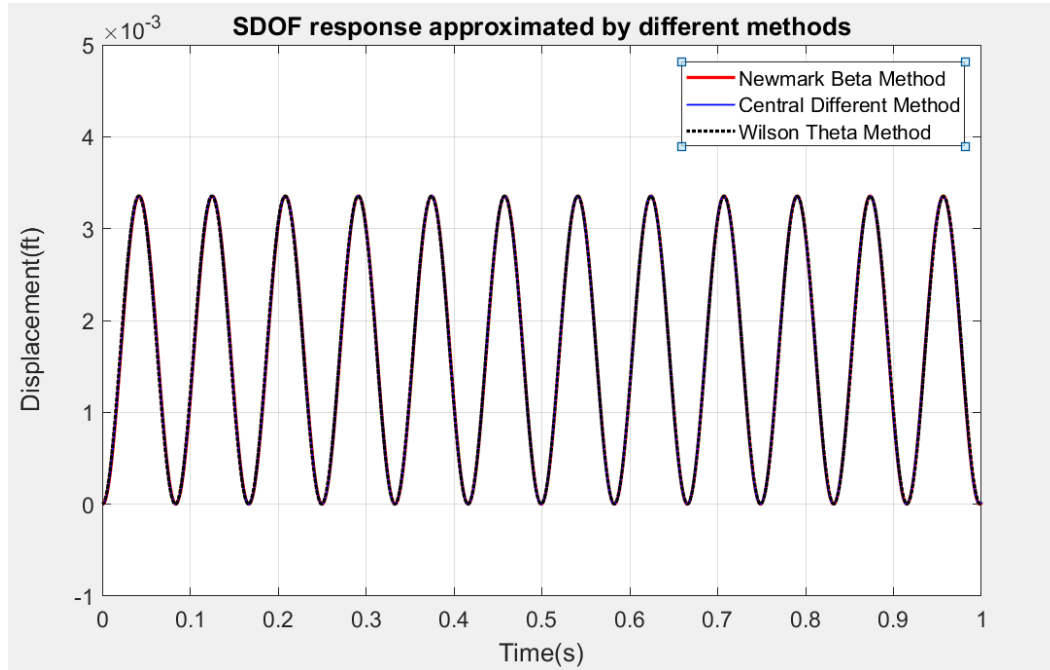
Figure 11. Plot shows the numerical results match when time step is small for free damping vibration in SDOF

The 1997 Accuracy of Response of Single Degree of Freedom report [12] discusses a ***period elongation*** phenomena where the displacement "lags" from each other as time step gets larger. To illustrate, the following shows the period elongation problem when the displacement was solved at time step Δt=0.01 seconds.
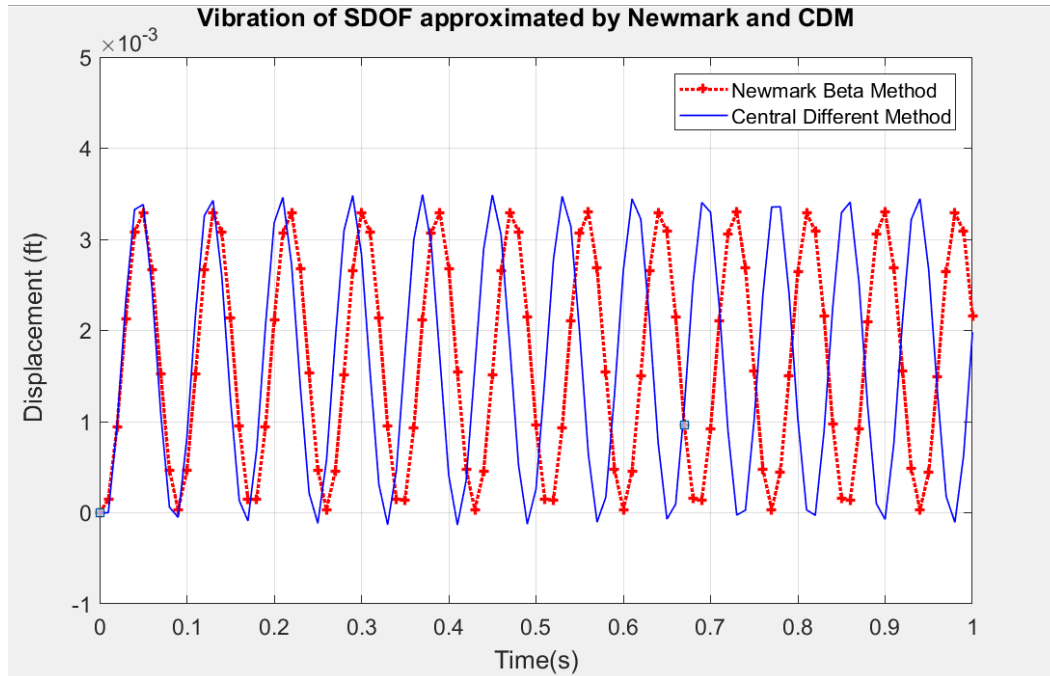
Figure 12. Period Elongation in Newmark beta method and CDM method

As seen in the figure above, the approximation lags away from each other when time step gets larger. Later in this report, we will discuss the stability of the result with respect to the time step in the analysis. The FEA modeling section will cover the check needed to control the for time step in modeling to avoid period elongation as well as to achieve convergence.

B.   Accuracy for spatial discretization in FEM

1.   Accuracy in mesh discretization and integration point selection within an element

This section presents a series of results for a cantilever beam study (Figure 14 to Figure 18). In this example, a beam is modeled with one fixed end. Lateral load is applied on the free end of the beam. Solid elements are used and a parametric study was done to show the sensitivity of the model with respect to the selection of mesh size and different number of integration points. The illustration of the various meshes considered in this study are shown in Figure 14 through Figure 18.
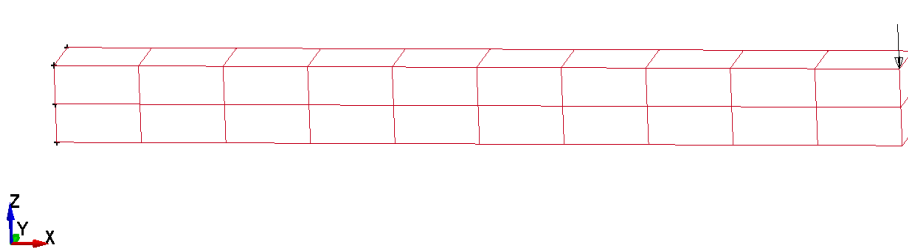
Figure 13. Beam bending test 1 with 20 solid 8 node brick elements
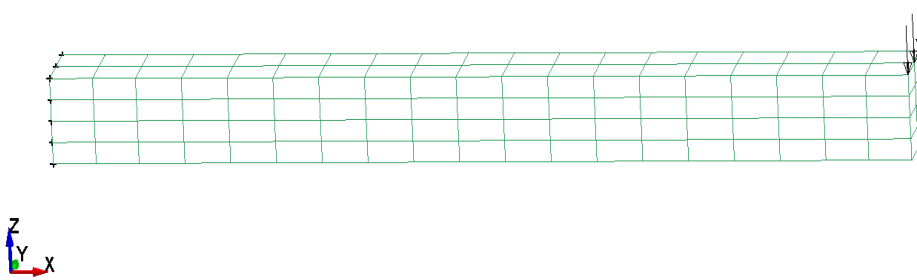
Figure 14. Beam bending test 2 with 152 solid 8 node brick elements
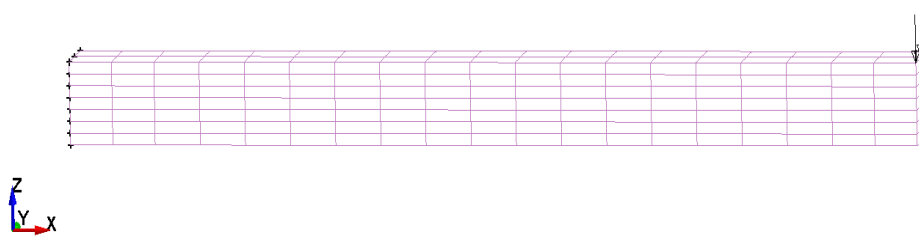
Figure 15. Beam bending test 3 with 266 solid 8 node brick elements
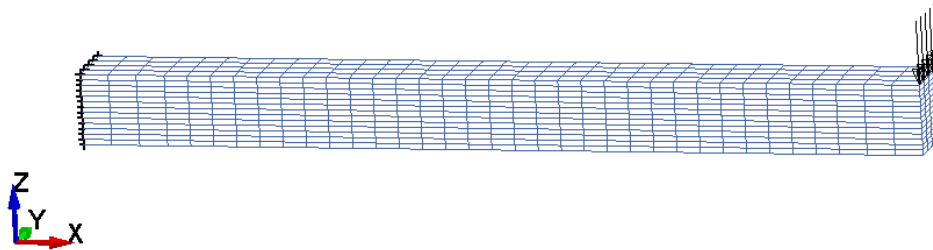
Figure 16. Beam bending test 4 with 1,820 solid 8 node brick elements
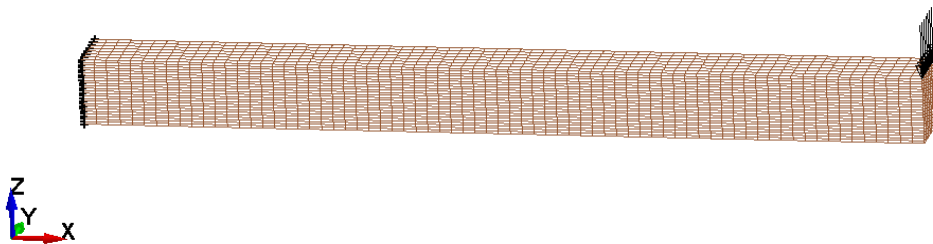
Figure 17. Beam bending test 5 with 12,425 solid 8 node brick elements

Table 8 lists the model geometry, load information, model mesh aspect ratio and the displacement determined from the analytical solution at the free end of the beam.

Table 8. Beam model geometry, load information, and the analytical solution

Length (in) x Width (in) x Height (in)	600.0 x 50.0 x 50.0
Load/Force at free end (lbs)	200.0
Young Modulus (lb/in^2)	29,000,000.0
Analytical deflection at beam end (in)	-0.0009534

The following Table 9 lists the displacement at the free end of the beam for the three solid elements (element aspect ratio of 3:1) obtained in LSDYNA analysis using implicit solving scheme for a static analysis.

Displacement at the free end of bending in beam example [inches]*

Table 9. Displacement [inch] at free end in beam example.

Discretization (number of elements)	Solid element type 1 (constant stress solid element)	Solid element type 2 (fully integrated solid)	Solid element type -2 (fully integrated element with modification to accounted for poor aspect ratio)
20	-0.00127	-0.000633	-0.000938
152	-0.00101	-0.000835	-0.000950
266	-0.000969	-0.000824	-0.000954
1,820	-0.000965	-0.000912	-0.000957
12,425	-0.000961	-0.000946	-0.000957

Note*: Displacements taken at the extreme fiber on the tension face of the cross section

1. Error of the approximation as compared to analytical results

Using the analytical displacement, errors are calculated for beam with different element and are listed as following Table 10.

Table 10. Differences in choices of elements

Discretization (number of elements)	Solid element type 1 (constant stress solid element)	Solid element type 2 (fully integrated solid)	Solid element type -2 (fully integrated element with modification to accounted for poor aspect ratio)
20	33.21%	33.60%	1.61%
152	5.94%	12.42%	0.35%
266	1.64%	13.57%	0.07%
1,820	1.22%	4.34%	0.38%
12,425	0.80%	0.77%	0.38%

One major factor in engineering FEA problem is the computational time. The following table lists the CPU time needed for each models.

1.1   Efficiency in mesh and integration point selection within an element in term of CPU/Analysis time

CPU/Analysis time, seconds

Table 11. Table shows CPU/Analysis time (in seconds) as number of elements in a model changes

Discretization (number of elements)	CPU/Analysis time, seconds		
	Solid element type 1 (constant stress solid element)	Solid element type 2 (fully integrated element)	Solid element type -2 (fully integrated element with modification to accounted for poor aspect ratio)
20	13	13	16
152	16	18	24
266	17	20	28
1,820	68	83	135
12,425	1278	1373	1350

Figure 19 plots the analysis time versus the number of elements with using the data from the table above.
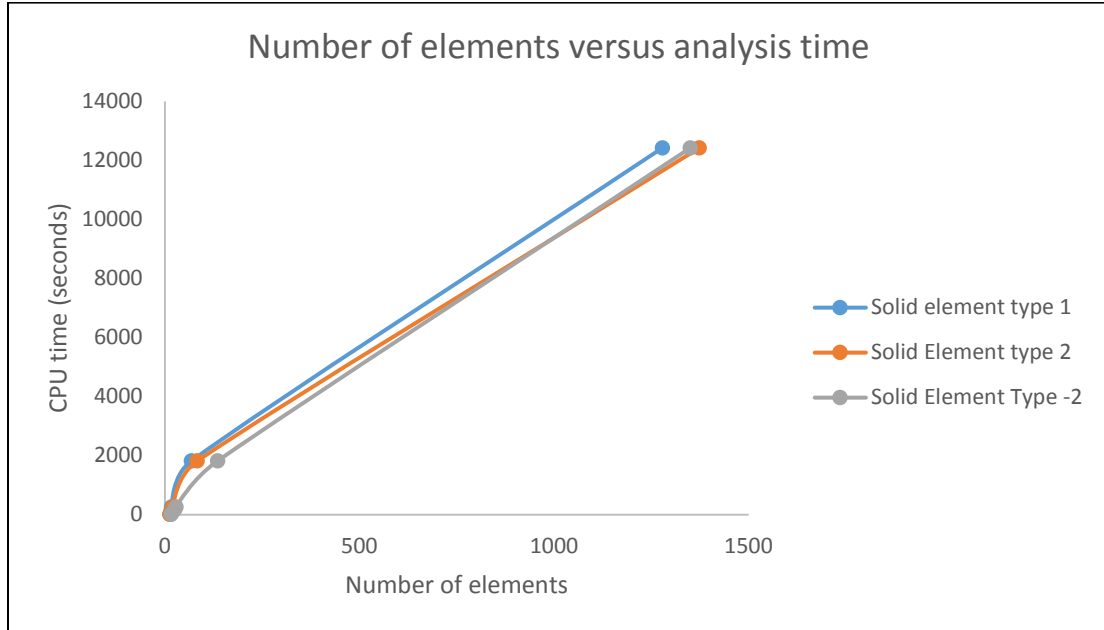
Figure 18. Number of elements versus analysis time

As seen in Table 11, the fully integrated element (solid element type 2) appears to be the "stiffest" one. This element assumes that pressure is constant throughout. This assumption intends to avoid pressure locking during nearly incompressible flow (stiff/dense flow type). This element type however, has the shear locking in its formulation and numerical response could be excessively stiff.

Table 12 shows the comparison in term of displacement result and CPU analysis time for solid element type 1 (constant stress, one integration point element and type 18 (formulated with 8 integration points and is recommended for linear statics only).

Table 12. Table shows displacement at free end of the beam, in inches, with respect to number of elements.

Discretization (number of elements)	Displacement at free end of the beam, inches, solved by implicit method	
	Solid element type 1 (constant stress solid element)	Solid element type 18 (8 point enhanced strain solid element for linear analysis only)
20	-0.00127	-0.00127
152	-0.00101	-0.00101
266	-0.000969	-0.000969
1,820	-0.000965	-0.000965
12,425	-0.000961	-0.000961

With the analytical displacement, the numerical errors are computed and are listed as following:

1. Error of the approximation as compared to analytical results

Table 13. Error in displacements versus number of elements in model

Discretization (number of elements)	Solid element type 1 (constant stress solid element)	Solid element type 2 (fully integrated element)
20	33.21%	33.60%
152	5.94%	12.42%
266	1.64%	13.57%
1,820	1.22%	4.34%
12,425	0.80%	0.77%

The following table lists the comparison in term of CPU/analysis time for the solid element type 1 (with constant stress) and for the modified solid element type 18 (8 integration point, recommended element type for linear analysis).

1. CPU/Analysis time, seconds

Discretization (number of elements)	Solid element type 1 (constant stress solid element)	Solid element type 18 (formulated for linear analysis only)
20	13	10
152	16	18
266	17	17
1,820	68	68
12,425	1278	1310

As seen in the tables above, the element with 8-integration point formulated for linear analysis gives good results. In a small model with less elements, the results appears to be even more efficient than the LS-DYNA default solid element type 1.

1.2 Error in term of solution scheme selection

Displacement of the beam at free end with different solution solver methods: Implicit versus Explicit

Table 14. Table shows the differences in displacement when using different solution solver methods

Discretization (number of elements)	Solid element type 1 (constant stress solid element) (Implicit analysis)	Solid element type 1 (LS-DYNA default element type) (Explicit analysis, time step of dt=0.9 seconds)
20	-0.00127	-0.00112
152	-0.00101	-0.00103
266	-0.000969	-0.00095
1,820	-0.000965	-0.000969
12,425	-0.000961	-0.000977

1.1 Error in displacement with different solution solver methods: Implicit versus Explicit

Table 15. Table shows the error in displacement with different solution solver methods.

Discretization (number of elements)	Solid element type 1 (constant stress solid element) (Implicit method)	Solid element type 1 (LS-DYNA default element type) (Explicit analysis, time step dt=0.9 seconds)
20	-33.21%	-17.48%
152	-5.94%	-8.04%
266	-1.64%	0.35%
1,820	-1.22%	-1.64%
12,425	-0.80%	-2.48%

Discussion of the solid element types and the mesh transition region in the FE models

2.1.   Element types

It is ideally to maintain the same type of element in the mesh, with knowing that the change in element configuration (mesh shape) essentially affects the result. Discussion of integration point of solid element types and mesh transition, we discussed the integration where "weighted average Gauss points" evaluated per selective surfaces among the element volume. Hence, the element stiffness varies as the mesh shape varies. In the following Table 16 one can see the errors are high as compared to the undistorted element elements.
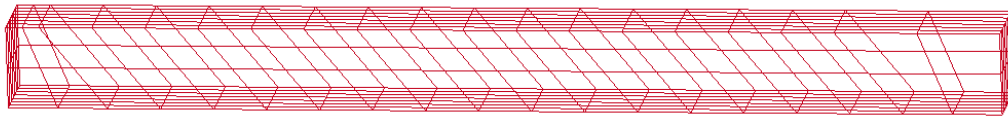
Figure 19. Figure shows mesh configuration changed

Table 16 shows the errors of the perfect cube mesh and the "distorted" mesh elements

Table 16. Difference in displacement at mesh configuration changes

Discretization (number of elements)	Solid element type 1 (LS-DYNA default element type) (Implicit analysis)	Solid element type 1 with mesh shape being changed (See Figure 22.)
20	-33.21%	4421%
152	-5.94%	33%
266	-1.64%	33%
1,820	-1.22%	7%
12,425	-0.80%	2%

In some cases, when tetrahedron elements are used, mesh density should be much denser as compared to the hexahedron elements. The following tables shows high error in displacement due to high element stiffness for tetrahedron elements.

Figure 20. 80 solid 6 node tetrahedron (triangular) elements
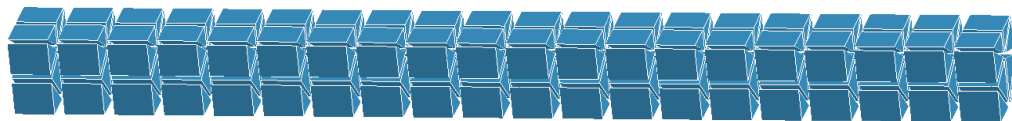
Figure 21. 160 solid 6 node tetrahedron elements

Figure 22. 320 solid 6 node tetrahedron elements

Table 17. Table shows the error in free end displacement for beam model with 6 node tetrahedron elements

Discretization (number of elements)	Solid element type 1 (LS-DYNA default element type) (Implicit analysis)
80	-57%
160	-80%
320	-27%

2.2   Time step needed when different element types are use

This discussion addresses the question of which time step should be used in the analysis. In a vibration/dynamic associated analysis, wave propagation velocity within an element is used to determine a time step. It should be noted here that the time step limit as "wave transit over the shortest element dimension" is applicable only to explicit integrator; in addition, a shortest element dimension is not necessarily the shortest element side. A continuum wave propagation in a 3 dimensional model is defined as following:

$$c = \sqrt{\frac{E(1-v)}{(1+v)(1-2v)\rho}}$$ (Eq.53)

Where:

E=Young's modulus of material

v=Poisson's ratio

ρ=specific mass density.

Critical time step for an analysis is defined as:

$$\Delta t = \frac{l}{c}$$ (Eq.54)

Where:

l= characteristic length of the smallest element in the model

As seen in the above equation, the smallest element in the model controls the time step and consequently controls the overall analysis time. The following table lists the evaluation of the characteristic length l_c of the three solid element types:

Table 18. Element types

Element type	Evaluation of characteristic length l
8 node solid brick element	$l = \dfrac{Volume\ of\ element\ V}{max\ area\ A\ of\ one\ side\ of\ the\ element}$
Tetrahedron 5 node solid element	l= smallest length of one of the element sides
Tetrahedron 6 node solid element	$l = 1/\sqrt{B_{ij}B_{ij}}$

3. Finite element solution errors

The 1982 Finite Element Procedure in Engineering Analysis prepared by K.J Bathe states: "Considering the approximate FE solutions to the exact responses of the mechanical idealization, we need to recognize that different sources of errors (that) affect the FE solution results". The following table lists the general source of errors:

Table 19. General sources of errors in FEA.

Type of problems (Sources: [4])	Errors occur in (Sources: [4])	Discussion and demonstration in
1. Discretization	Use in finite element interpolation	Section III. part C.
2. Numerical integration	Evaluation of element stiffness with respect to numerical integration point	Section III. part C.
3. Evaluation of constitutive relations	Uses of nonlinear material models	Readers are referred to [1]
4. Solutions of dynamic equilibrium equations	Direct time integration, mode superposition	Section III. part C.
5. Solution of FE equation by iteration	Newton Raphson, Newton-like methods, Eigen-solutions	Section II. part 2.
6. Time step	Time step affects greatly in truncated error. Especially for stability in explicit method.	Section IV. part A.

V. Development of FE models

There are two ways TSC constructs FE models of a dam-foundation-reservoir system using TrueGrid and LS-PRPOSTsoftware.

A. Modeling with TrueGrid

TrueGrid is currently used by the TSC in development of the most dam models. Using TrueGrid, a geometry and mesh of a dam structure is created using the built-in mesh generator. TrueGrid mesh generator engages users with 3 dimensional multiple block meshes, surface, and curves. Each block in TrueGrid is defined with indices/numbers according to the local i,j,k coordinate system. Once a model is constructed, an output file with geometry, nodes, element connectivity, material properties, and loads is saved and exported into LS-DYNA. The computations and the result post-processing in done in LS-PREPOST (LSDYNA). The benefit of using TrueGrid mesh generator software is that the users have control of a series of commands in generating block meshes. In some cases, when the geometry of dams (or other structures) are similar, certain "set" of commands could be reused, and thus provide benefits in mesh/modeling. A dis-advantage of using TrueGrid mesh generator is when complex configurations need to be modeled. Essentially, the script coding for certain irregular shapes or complicated geometry becomes time consuming if not inefficient. A selected tutorial for this modeling approach is included in Appendix 1.

B. Modeling with LS-Prepost

This section describes an approach to develop the FE model using the built-in LSDYNA Prepost features. The LS_PPREPOST allows the creation of a geometry and mesh of the model, performs the FE analysis and then post-processes the results inside the LSDYNA software. Graphical interface as well as scripting commands (not as user friendly as compared to TrueGrid) can be used in developing the FE model. The modeling portion with multiple modeling/visualization features is similar to what used in drafting (i.e AutoCAD). When dealing with a complicated/sophisticated model, the users could manually select mesh shape for a particular element, or can generate an element directly. In addition, users can utilize a draft input file created via drafting software such as AutoCAD, IPES, or any other drafting program and import the geometry of the model into LS-Prepost. In engineering practice, when drawing/drafting tasks are crucial, the drawing files could be reused, this benefits the project time. A selected tutorial for this modeling approach is included in Appendix 2.

VI. Conclusions

This report summarizes our work for the Dam Safety funded research project
"Guide for Analysis of Concrete Dam Structures using Finite Element Methods."
Here we discuss fundamental topics such as the mathematical formulation in term
of both field partial differential equations and variational principles, boundary
conditions for dam and fluid-dam interaction, the matrix formulation of the FEA
equations, and solution approximation schemes for dams excited by the seismic
events. Two major time integration method classes in dynamic analyses: explicit
and implicit methods are covered and illustrated with algorithm steps.

We have primarily focused on topics that affect numerical stability and accuracy.
These include: the effect of time step on numerical stability, sensitivity of results
to mesh fineness and element-level numerical integration schemes, and degradation
of accuracy due to element type and mesh distortion. These topics span a wide
range from physical and mathematical modeling, passing through FE model
discretization, and getting to numerical results.

To help FEA model preparation, the report specifically covers procedures used by
TSC modeling software using TrueGrid and LSPREPOST (See Appendix 1 and 2).

The following are candidates to address in the future:

(1) Accounting for fluid-structure interaction (FSI) when the dam-reservoir system
may experience seismic events that can amplify damaging and destructive effects,
for examples cavitation shock or foundation slippage. Case studies that show FSI
effects are covered in [4].

(2) Estimating and assessing error of FEA solutions. The report briefly discusses
several modeling choices that directly affect accuracy: solution scheme, time step
size, mesh configuration and fineness, and element numerical integration. A recent
publication [13] address the topic of a posteriori error estimation using a more
mathematically oriented approach that takes into account various error sources in
practical and comprehensive manner. This integrated approach may be worthwhile
future investigation.

References

[1] J. Salamon, "Evaluation of Non-linear Material Models in Concrete Dams using Finite Element Analysis," U.S Department of the Interior, Denver, 2014.

[2] R. Koltuniuk , P. Percell and B. Mills-Bria, "State-of Practice for the Nonlinear Analysis of Concrete Dams 2013," U.S. Department of the Interior, Bureau of Reclamation, Technical Service Center, Denver, 2013.

[3] H. M. Westergaard, "Water Pressures on Dam during Earthquakes," American Society of Civil Engineers, vol. 98, no. 2, pp. 418-433, 1933.

[4] K.-J. Bathe, "Finite Element Procedures in Engineering Analysis," Prentice-Hall, Inc., Englewood Cliffs, 1982.

[5] J. Salamon and J. Manie, "Numerical Assessment of Hydrodynamic Loads Induced during Seismic Interaction between Reservoir and Concrete Dam," in United States Society on Dams (USSD) 2017 annual conference, Anaheim, 2017.

[6] C. Felippa, "Advanced Finite Element Methods," University of Colorado Boulder, Boulder, CO, 2013.

[7] "MIT Open Course Ware topic on Finite Element Procedures for Solids and Structures," MIT, Cambridge.

[8] S. E. Leon, G. H. Paulino, A. Pereira and I. F. Mennezes, "A unified Library of Nonlinear Solution Schemes," ASME Journal-The American Society of Mechanical Engineers, vol. 64, no. 040803, pp. 1-5, 2011.

[9] LS_DYNA Keyword User's Manual, Livermore: Livermore Software Technology Corporation, 2003.

[10] D. Roylance, "Finite Element Analysis," Department of Material Science and Engineering Massachusetts Institute of Technology, Cambridge, 2001.

[11] R. Regueiro, Introduction to the Finite Element Method, Denver: University of Colorado Boulder, 2011.

[12] R. M. Ebeling, R. A. Green and S. E. French, "Accuracy of Response of Single Degree of Freedom," U.S Army Corps of Engineers, Washington, 1997.

[13] T. Gratsch and K.-J. Bathe , "A posteriori error estimation techniques in practical finite element analysis," ELSEVIER Computers & Structures Journal, 2004.

[14] A. K. Chopra, Dynamics of Structures-Theory and Applications to Earthquake Engineering, Berkeley: Prentice Hall of India, 2008.

[15] Salamon, Jerzy; Manie, Jonna, "Numerical assessment of hydrodynamic loads induced during seismic interaction between reservoir and concrete dam," in Unite States Society on Dams 2016 Annual Conference, Anaheim, CA, 2016.

Appendix

Appendix 1 - Modeling approach using TrueGrid

```
c
c      Seismic Incuded Loads on Dams
c
title Seismic Incuded Loads on Dams: Jerzy Salamon, July 2012
c
c      DEFINITION OF GEOMETRY PARAMETERS
c      ==================
      csca 12.0
c  interrupt
c
para
c ====Small gravity Dam ==========
      H               100.0              c RWS Depth [FT]
      h1              20.0               c Chimney Depth [FT]
      da        0.0                 c U/S Toe offest
      db        96.0                  c Dam Width at Base
      dc        40.0                  c Dam Width at Crest
      dd        40.0                  c Dam Width at chimney
bottom
      lx              5                  c Reservoir Length
      x Hight
      th              1.0                c Thickness of
Reservoir Stip
{
c ====Konya Dam ==========
      H               338.0              c RWS Depth [FT]
      h1              120.0              c Chimney Depth [FT]
      da        9.08                c U/S Toe offest
      db        230.28                  c Dam Width at Base
      dc        48.5                  c Dam Width at Crest
      dd        66.86                  c Dam Width at chimney
bottom
      lx              10                 c Reservoir Length
      x Hight
      th              10.0               c Thickness of
Reservoir Stip

c ====Pine Flat Dam ==========
      H               400.0              c RWS Depth [FT]
      h1              50.0               c Chimney Depth [FT]
      da        16.75                c U/S Toe offest
      db        314.0                  c Dam Width at Base
      dc        32.0                  c Dam Width at Crest
      dd        33.25                  c Dam Width at chimney
bottom
      lx              10                 c Reservoir Length
      x Hight
      th              10.0               c Thickness of
Reservoir Stip
}
      n               1                  c  Global Mesh factor
      ny2       9                        c Water & dam vert
```

1

```
Mesh at chimney
      ny1       39                             c Water & dam Mesh
below chemney
      nxb       9                              c Mesh at Dam Base
      nxf1      49                             c Mesh at reservoir
far from dam
      nxf2      99                             c Mesh at reservoir
close to dam
;
      c
c
c     ==========================
c     DEFINITIONS OF DAM GOEMETRY
c     ==========================
block      1 2  ; 1 2 3 ;  1 2;
           0  %db; 0 [%H-%h1] %H;  0 %th;
c
     pb 1 2 1  1 3 2 x %da
     pb 2 2 1  2 3 2 x [%dc+%da]
     pb 2 2 1  2 2 2 x [%dd+%da]
c
           mate 1
c          mti ;2 4;; 1          c dam
c
           mseq i  %nxb;
           mseq j [%n*%ny1] [%n*%ny2] ;
c          mseq k %nk;
c
           bi ; ;-1 ; dz 1 ;
           bi ; ;-2 ; dz 1 ;
c    bi   ; -1 ;; dx 1 dy 1 dz 1;

           bi ;-1 ; ;  dy 1 ;
c
nseti ; -1; ;= 11
C interrupt
     endpart
c
c     ==========================
c     RESERVOIR
c     ==========================
block      1 2 3 4; 1 2 3 ;  1 2;
           [-%H*(%lx+0.5)] [-%H*%lx] [-%H*2.0] 0 ; 0 [%H-%h1] %H;
0 %th;
C

c          mate 2
           mti 1 2;;; 51          c PMF
           mti 2 4;;; 2           c Reservoir
c
           mseq i 7 [%n*%nxf1] [%n*%nxf2];
           mseq j [%n*%ny1] [%n*%ny2];
```

```
c          mseq k %nk;
c
c     bi -1 ; 1 3;; dx 1 ;
      bi 2 4; ;-1 ; dz 1 ;
      bi 2 4; ;-2 ; dz 1 ;
      bi ;-1 ; ; dy 1 ;
c PMF
      bi 1 2; ; ; dx 1 dy 1 dz 1;
c
      endpart
c
{
c     ==========================
c     Foundaiton
c     ==========================
block      1 2 3 4 5; 1 2  ;  1 2;
           [-%H*%lx] [-%H*2.0] 0 %db [%H+%db]; [-2.0*%H] 0 ;  0 %
th;
c
           mate 3
c                    mate 3
c
      bi -1 ; 1 2;; dx 1 ;
      bi -4 ; 1 2;; dx 1 ;
      bi  ; -1 ;; dx 1 dy 1 dz 1;
      bi ; ;-1 ; dz 1 ;
      bi ; ;-2 ; dz 1 ;
c
           mseq i [%n*%nxf1] [%n*%nxf2] %nxb 5 ;
           mseq j 10 ;
c interrupt
      endpart
c
}

c     ###############################################
c     MERGE THE BLOCKS AND CREATE NODES AND ELEMENTS
c     ===============================================
c
      merge
      stp 0.005
c
c     VERBATIM
c     ========
c
verbatim
$
$     END THE VERTATIM CARDS
$     ======================
endverbatim
c
c     SET A FLAG TO CREATE AN LS-DYNA INPUT FILE AT THE NEXT
```

```
WRITE COMMAND
c
=====================================================================
===
echo Specify LSDYNA as the output format
lsdyna keyword
c
c     WRITE AN OUTPUT FILE
c     ====================
echo Write out an LSDYNA input file (.k) called trugrdo
write
```

```
$# LS-DYNA Keyword file created by LS-PrePost 3.1 (Beta) -
28Jan2011(13:51)
$# Created on Jan-6-2012 (15:06:03)
$ *KEYWORD
*KEYWORD memory=99999000 ncpu=4
*TITLE
$# title
$ ========================================================
Time analysis of dam-foundation system
$
$      CONTROL JOB TERMINATION
$      =======================
$      Set the program to run for xx.xx seconds
*CONTROL_TERMINATION
   0.67, 0, 0.000E+00, 0.000E+00, 0.000E+00
$
$      SET THE TIMES WHEN D3PLOT FILE SAVED
$        (Set to 0.5 for Static and 0.01 for Dynamic)
$      ===================================
$      Write data to the d3plot files to create plots at every 0.1
seconds
*DATABASE_BINARY_D3PLOT
   0.001
$
$ *DATABASE_HISTORY_NODE
$#      id1         id2         id3         id4         id5         id6
id7         id8
$       101          56        2471        4746           0           0
0           0
$ *DATABASE_HISTORY_SOLID
$#      id1         id2         id3         id4         id5         id6
id7         id8
$        81          85          89          93          97         233
237         241
$       245         249           0           0           0           0
0           0
$ *BOUNDARY_SPC_SET
$#     nsid         cid        dofx        dofy        dofz        dofrx
dofry       dofrz
$         1           0           1           1           1           0
0           0
$ *SET_NODE_LIST
$#      sid         da1         da2         da3         da4      solver
$         1       0.000       0.000       0.000       0.000
$#     nid1        nid2        nid3        nid4        nid5        nid6
nid7        nid8
$     30973       30974       30975       30976       30977       30978
30979       30980
$ ==============================
$ PART 1 - DAM
$ ==============================
*PART
```

1

```
$# title
Dam
$#      pid     secid       mid     eosid      hgid      grav
adpopt      tmid
          1        18         1         0         0         0
0          0
*SECTION_SOLID
$#   secid    elform       aet
         18        -2         0
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db   not
used
          1   2.32e-4   4.00E+6   0.20000     0.000     0.000
0
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2
qb/vdc        qw
          1         5     0.000         0     0.000     0.000
0.000     0.000
$ ==============================
$ PART 2 - WATER
$ ==============================
*PART
$# title
Reservoir
$#      pid     secid       mid     eosid      hgid      grav
adpopt      tmid
          2        19        12         0         2         0
0          0
*SECTION_SOLID
$#   secid    elform       aet
         19         0         0
$
$    ELASTIC_FLUID MATERIAL MAT_001
*MAT_ELASTIC_FLUID
$#     mid        ro         e        pr        da        db
Bulk K
         12   0.94e-4       3E+5  0.480000     0.000     0.000
3e+5
        0.2      1E+20
$
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2
qb/vdc        qw
          2         5     0.000         0     0.000     0.000
0.000     0.000
$ ==============================
$ PART 3 - FOUNDATION
$ ==============================
*PART
$# title
Foundation
```

```
$#      pid      secid       mid      eosid      hgid       grav
adpopt     tmid
         3         3         3         0         0         0
0         0
*SECTION_SOLID
$#   secid    elform       aet
         3         1         0
*MAT_ELASTIC
$#      mid        ro         e        pr        da        db  not
used
         3   4.4e-14     3E+16  0.200000     0.000     0.000
0
$ ===============================
$ PART 4 - PML
$ ===============================
*PART
PML for impounded water
51,51,51,0,51,0,0,0
$
$ DEFINITION OF MATERIAL    51
$
$ *MAT_PML_NULL
$ 51,0.00009353,0.0
*MAT_PML_ELASTIC_FLUID
51,0.00009353,300000,0.0
*HOURGLASS
51,0,0.0,0,0.0,0.0
*SECTION_SOLID
51,2
$
$ ===============================
$
$     APPLY SEISMIC LOAD
$     =================
$ *BOUNDARY_PRESCRIBED_MOTION_RIGID
$ PID DOF VAD LCID SF VID DEATH BRITH OFFSET1 OFFSET2 MRB NODE1
NODE2
$  displacement
$ 1,1,2,1,-0.01,0,13
$    acceleration
$ 20,1,1,1,-38.64,0,13
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$$$$$$$$$$$$$$$$
$
$$$  Prescribe the velocity of the nodes on one end of the plate.
$
*BOUNDARY_PRESCRIBED_MOTION_SET
$      nid       dof       vad      lcid        sf       vid
        11         1         1         1    -38.64         0
$
$
```

```
$*SET_NODE_LIST
$       sid
$ 4         1
$     nid1      nid2      nid3      nid4      nid5      nid6
nid7      nid8
$ 4       231       232       233       234       235
$
$
$      INCLUDE LOAD CURVES
$      ============================
*INCLUDE
$sine-out_0-10
$ sine-out_0-16
sine-out_0-66
$ sine-out_1-3
$ sine-out_100
$
$      INCLUDE GEOMETRY AND BOUNDARY
$      ============================
$
*INCLUDE
trugrdo
$
*END
```
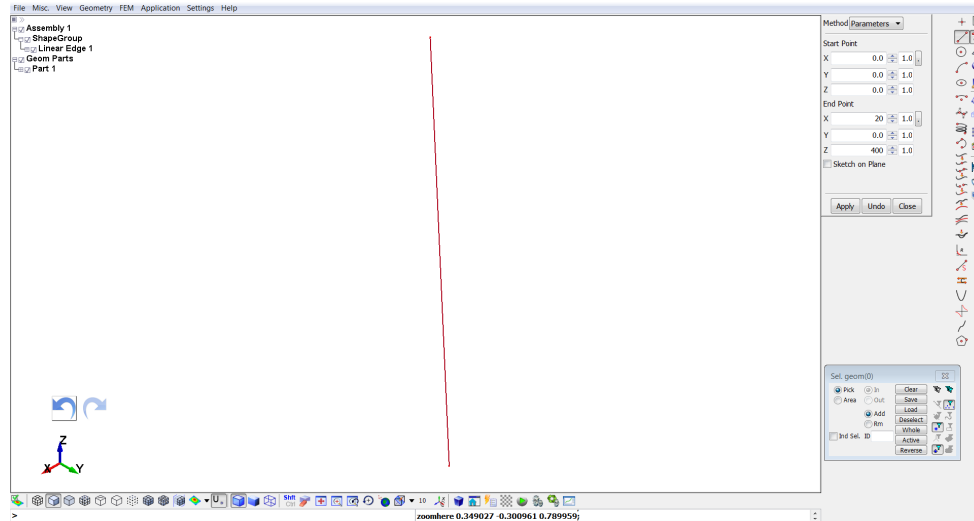
Appendix 2 - Developing concrete dam models/examples

The following example provides a procedure of how to create a solid block mesh for a gravity dam configuration using LSPREPOST.

A. Construct geometry and generate elements

1. Create a plane with (4) sides using curve: Geometry→Curve→Line→Method: Parameters→Input all point coordinates→Apply

2. For the arc portion:
   Geometry→Curve→Circular Arc→ In this step, use a drop down menu in the Method: Two points/Radius

   Use general selection box, frame two ends of the vertexes / which will select two points→ Enter 84 feet for Radius→ Apply→Close. Delete the extra arc curve

3. Create a surface off of the N-line plane: Geometry→Surface→N-Side Surface→Method→Continuity→Edge list→All the edges in the plane→ Apply→Done

4. Create a shell from the surface: Geometry→Surface→N-side Surface

5. Mesh the shell: FEM→Element and Mesh→N-Line Mesher (input the number of element on each line)

6. Generate a block of solid using the shell mesh as a seed plane: Element and Mesh→Element Generation→Solid→Shell Drag→Enter the thickness (in this example, it is the section thickness of the dam block in the cross canyon direction),→Enter the segment(element number through thickness→Enter the normal vector (indicating the direction of the thickness axis (0,1,0) in this example)→Create→Done

*Note: because LSDYNA generates solid elements based on the shell element skeleton, there will be a shell part (also known as seed plane) that was built in the model. (This can be seen in the tree menu (See LSHELL2 and LSHELL3 in figure xx). This seed plane is not needed, this needs to be removed once the model is completed.
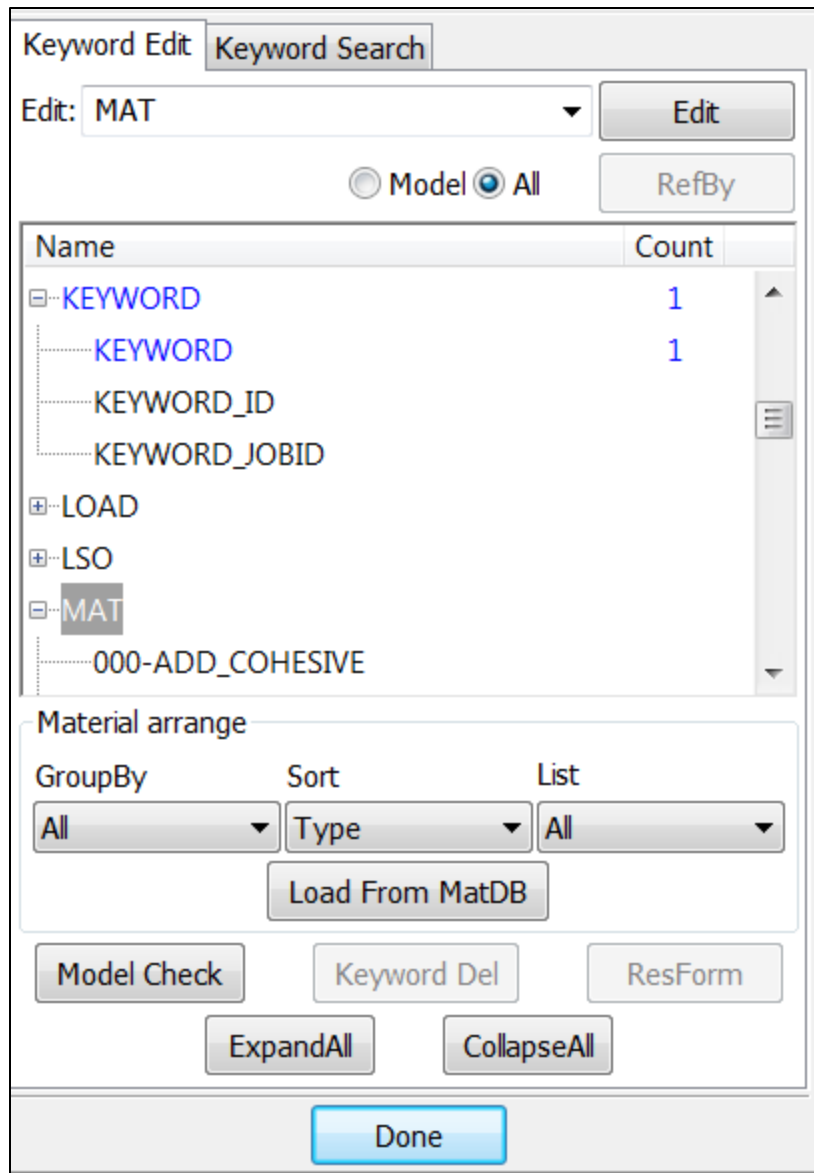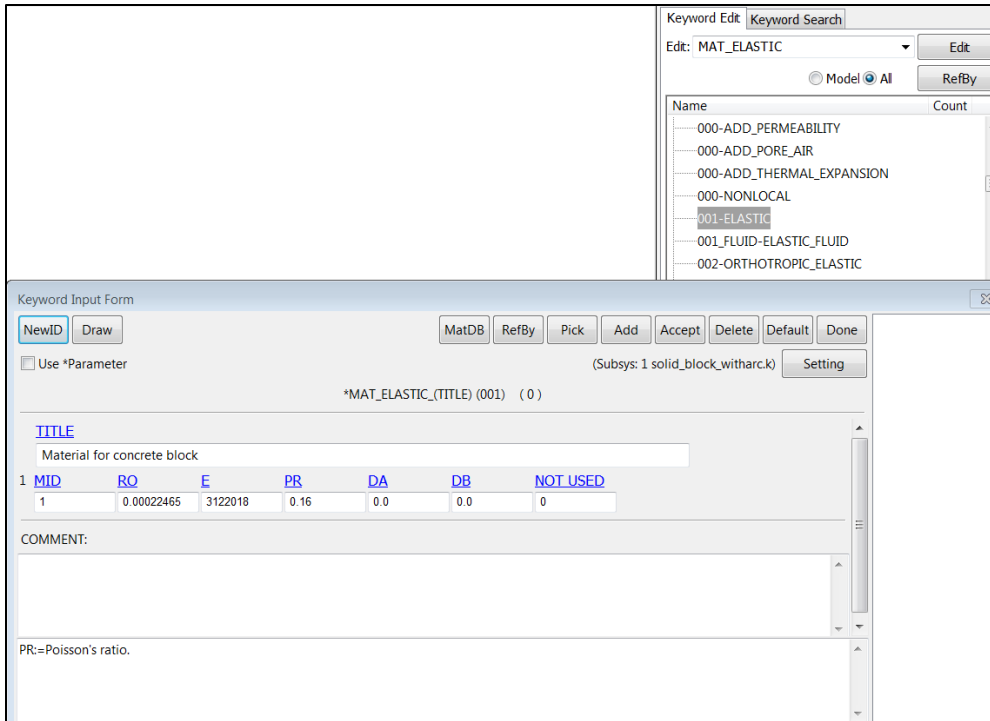
B. Define material

1. FEM→Model and Part→Keyword Manager→MAT→Make sure in Keyword Edit, check the option All

2. Expand MAT option→Select 001-ELASTIC

TITLE: put in title for the block model, this is optional.

MID: Material identification. A unique number has to be chosen.

RO: Mass density. (Please see remarks note 1)

E: Young Modulus. In this example, E was calculated for $f'c$ of 3,000.00 lb/in^2 for concrete.

PR: Poison ratio

DA: Axial (force) damping factor (used for Belytschko-Schwer beam element type 2 only). (Please see Note 2)

DB: Bending (moment) damping factor (used for Belytschko-Schwer beam element type 2 only). (Please see Note 2)

Note:

1. *As discussed in the time integration equation of motion section in the report, LS-DYNA solves the system of equation of motion with respect to time. Displacement, velocity, or acceleration solved with using the input data. In LS-DYNA, there is no way of identifying what units the model has. At all time, the modeling engineers/analysts are opted to keep track of the units and ensure the units are compatible. Couple useful ways to maintain the compatibility in unit is to check:*

$$1(force\ unit) = 1(mass\ unit) \times 1\ (acceleration\ unit)$$

$$1\ (acceleration\ unit) = \frac{1\ (length\ unit)}{[1(time\ unit)]^2}$$

The   force   equation   then   becomes:

$$1(force\ unit) = 1(mass\ unit) \times 1\frac{1\ (length\ unit)}{[1(time\ unit)]^2}$$

$$1(mass\ unit) = 1(force\ unit) \times 1\frac{[1(time\ unit)]^2}{1\ (length\ unit)} = 1(slug\ unit)$$

Since:

$$1(slug\ unit) = 1(lbf) \times 1\frac{[1(time\ unit)]^2}{1\ (length\ unit)}$$

This explains the unit of the mass density in LS-DYNA card being written in a slug form as following:

$$1(mass\ density\ unit) = \frac{1(mass\ unit)}{1[(length\ unit)]^3} = \frac{1(slug\ unit)}{1[(length\ unit)]^3}$$

Therefore as seen in this example, the input for mass density in concrete was calculated as (150 lb/ft³)/(32.2 ft/s²)/(12⁴ (ft⁴/in⁴))=0.00022465
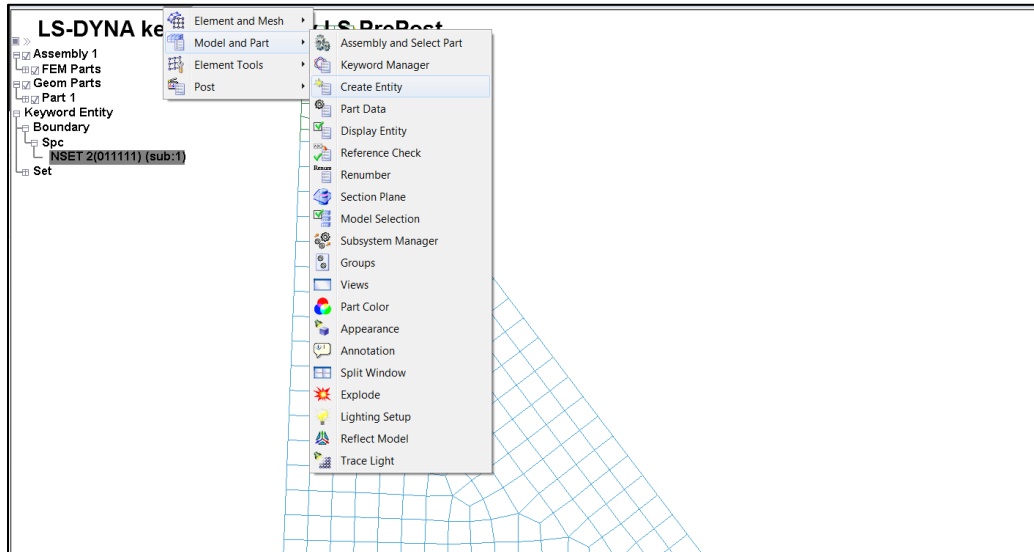
2. *The axial and bending damping factors are used to damp down numerical noise. Basically, the update of the force resultants, F_i, and the moment resultant M_i, includes the damping factors:*

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right) * \Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right) * \Delta M_i^{n+\frac{1}{2}}$$
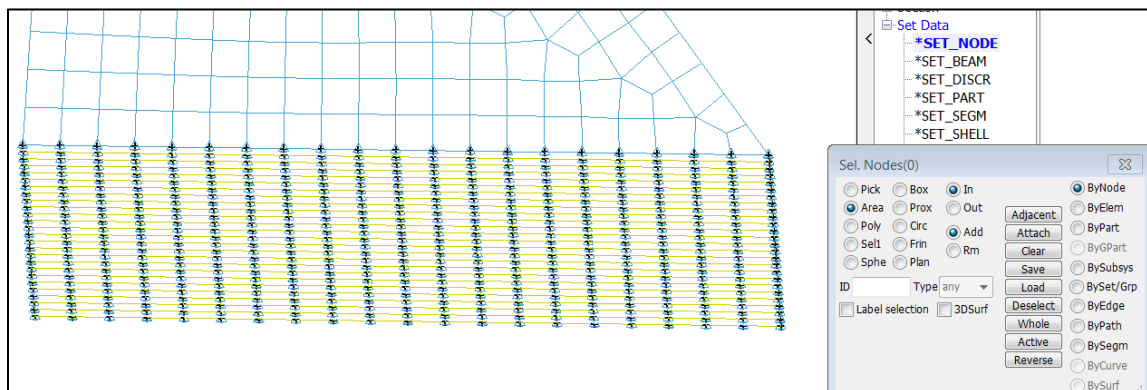
C. Define boundary condition:

1. FEM→Model and Part→Create Entity

In this example, a Set of nodes is defined at the bottom of the dam block assuming that the dam block is free in movement along x axis only.

2. Under Set Data→*SET_NODE→Select Cre (Create)→Input a unique identification number for the SET→Use a pop out Sel.Nodes window to select the nodes at the bottom of the dam.

3. Under Boundary→Spc→Select Cre (Create)→Select the Set that we just defined→Uncheck X displacement and check Y, Z, RX,RY,RZ
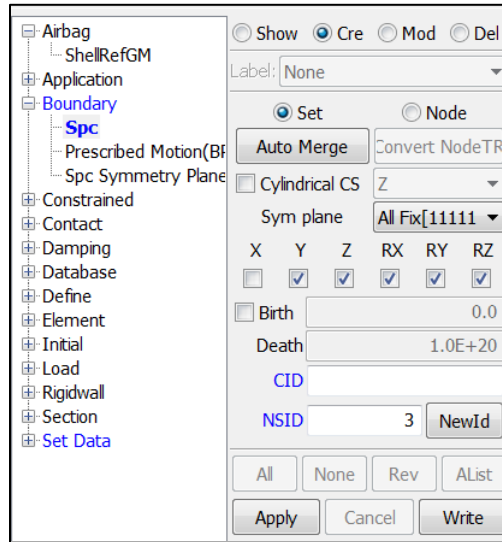

4. Apply load on the structure

Create a set of nodes at the US face of the block where the pressure is being applied.

5. Identify the analysis type / Select approximation scheme. Note that this portion could be done using keyword input.

6. Determine the time step needed for the analysis.

7. Check analysis. This option is an advantage of using LSPrepost. Prior to submitting the analysis, the input data can be checked.

   FEM→Model and Parts→Keyword Manager→Select Model Check
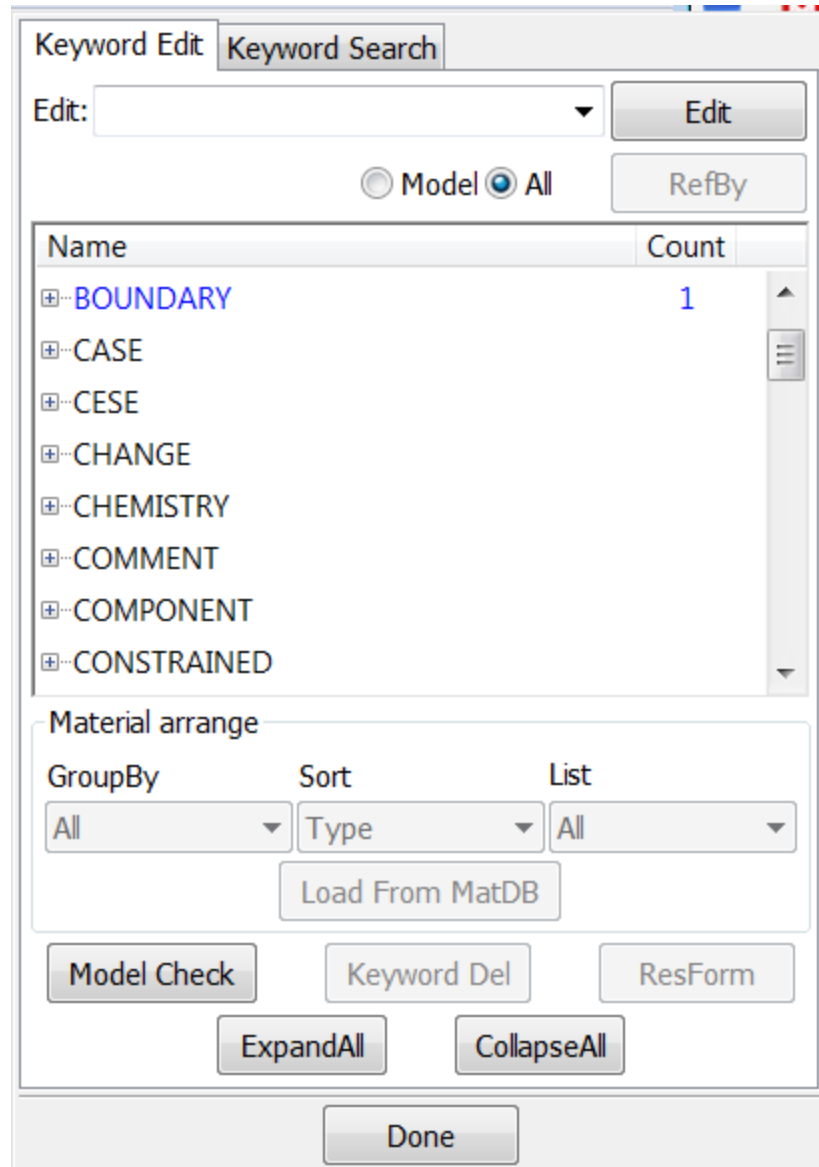
8.

Open Keyword Check

In this example, there are two seed shell planes used when building the solid elements, so one can see that the errors are at the PART.

| Element Quality | Keyword Check | Contact Check | Model Check Setting |

Total	Error(3)	Warning(899...	UnRef(2)	UnDefined(...
⊞ BOUNDARY(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ ELEMENT(7774)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ KEYWORD(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ LOAD(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(1)
⊞ NODE(8996)	Error(0)	Warning(8996)	UnRef(0)	UnDefined(0)
⊞ PART(3)	Error(3)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ SET(4)	Error(0)	Warning(1)	UnRef(2)	UnDefined(0)
⊞ TITLE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)

☑ Do not Check Contact   [Recheck]   [Model Clean]   [Write]   [Snap]

[Done]