SINGLE-DEGREE-FREEDOM PLASTIC ANALYSIS (SPAN32)

William F. Seipel, P.E.[1]
Scott Thomas, [2]
Timothy C. Knight, P.E.[3]

Abstract

SPAn32 is the product of many years of development. It was created as an in-house Single-Degree-of Freedom code to aid in the design and analysis of structural members subjected to dynamic loads. SPAn32 was written originally for design and analysis of hardened structural members subjected to high explosive loading. These dynamic loads can be generated from any type of explosion that results in a uniform load on the face of the member. The uniform loading could be (any of the following types;) airblast, ground shock, or any other transient uniform loading. SPAn32 creates the equivalent single degree of freedom model and calculates the dynamic response of a structural member. It is a useful tool in analysis of conventional construction when the structural member resistance function is of a bilinear form (elastic, elastic-plastic, and perfectly plastic) and the loading it is subjected to is any spatial uniform dynamic load. SPAn32 is currently in version 1.3.0.0 and is limited distribution, critical technology.

Current capabilities of the SPAn32 include these member model types: 1) one-way and two-way steel members, 2) one-way and two-way reinforced concrete members, and 3) a user-defined member bilinear resistance. SPAn32 has several special analysis features including, 1) membrane resistance function for laterally restrained reinforced concrete (fixed support conditions only), 2) concrete tension dynamic increase factor, 3) auto-calculation of dynamic increase factor, 4) strain hardening of steel, and 5) a deep beam model. These enhancements to SPAn32 more closely model the member response in a highly dynamic environment. Originally written for hardened construction subject to conventional weapons, the same SDOF principles and physics apply for convention construction subject to terrorist threat. A summary of SPAn32 is discussed and presented.

Introduction

The Single-degree-of-freedom Plastic Analysis (SPAn32) program is written in FORTRAN 77/90. It is a 32-bit program, which essentially means it written to take advantage of the Microsoft Windows 95/NT/2000/XP operating systems. SPAn32 exists in three different forms; as a Windows™ program with a windows interface, a character based DOS program, and a Dynamic Link Library (DLL). It is part of the Protective

[1] U.S. Army Corps of Engineers Protective Design Center, 12565 West Center Road, Omaha, NE 68144-3869, William.F.Seipel@usace.army.mil.
[2] U.S. Army Corps of Engineers Protective Design Center, 12565 West Center Road, Omaha, NE 68144-3869, Scott.W.Thomas@usace.army.mil.
[3] U.S. Army Corps of Engineers Protective Design Center, 12565 West Center Road, Omaha, NE 68144-3869, Timothy.C.Knight@usace.army.mil.

Structures Automated Design System (PSADS) collection of support programs and is used as a DLL in Automated Designer.

The SPAn32 program has a history of about 25 years, beginning in 1980 as SDOF. Over the years it has changed, being influenced by CBARCS, TM 5-1300, Structures To Resist Accidental Explosions (1969,1990), TM 5-855-1, Fundamentals of Protective Design For Conventional Weapons (1986), and more recently the Design and Analysis of Hardened Structures to Conventional Weapon Effects DAHSCWE UFC-3-340-1. In the early 1990s the program changed its name from SDOF to SPAn to differentiate SPAn from the common acronym SDOF. In 1994 as part of the PSADS development, SPAn was rewritten with a windows interface and the name changed to SPAnW. Later the name was changed to SPAn32 to reflect that it is written for Windows NT, 2000, XP 32 bit operating systems. SPAn32 continues to improve and find use in the design of conventional structures designed for blast load environments.

SPAn32 follows classical SDOF methods and guidelines given in the UFC 3-340-1 DAHSCWE manual chapters 10 and 11. The reinforced concrete model in SPAn32 follows closely with the guidance provided in American Concrete Institute (ACI) 318 Building Code. The Steel model in SPAn32 follows closely with the America Institute for Steel Construction (AISC) and TM 5-1300. The focus of this paper is to concentrate on the special features that have been incorporated into SPAn32 and a comparative analysis with DYNA3D.

Nomenclature

In using SPAn32 it is helpful to understand the language that is used to describe the program features, which are a combination of structural and weapon effects disciplines.

Units: SPAn32 will accept input in either US Customary (US) or System International (SI) units. SPAn is written to be unit friendly, the units are clearly marked for all input and output. When reading in a load file (pressure, or free-field velocity) the user has many unit possibilities not only SI or US but also seconds or milliseconds. There is one special case in the mass units which is due to customary representation, in the US units density is weight density, in SI units density is mass density; SPAn32 accounts for this difference.

Ultimate Resistance: Ultimate unit resistance (r_u) of one-way members is calculated using the full plastic hinge failure configuration appropriate to the member support condition. Ultimate unit resistance for two-way members is calculated using the TM 5-1300 yield line location charts and ultimate resistance equations for symmetrical cases. Moment capacities used in calculating r_u are the full Ultimate Moment (M_u) throughout the member except a reduced capacity of 2/3 M_u is used near each corner of the member where adjacent supports meet, as illustrated in Figure 3-3 of TM 5-1300.

Resistance Deflection Function: The resistance deflection function is a piecewise graphical representation of the strength and stiffness properties of the member. The resistance functions used in SPAn32 are base on yield-line theory and are a function of

the flexure response only. The SPAn32 offers several options in the development of the resistance deflection function used in the dynamic analysis see Table 1. These options will have an effect on the resistance function use in the SDOF calculation. There are several general resistance function types these include bilinear, multi-step, user defined bilinear and a membrane enhanced (compression and tension effects) options. In the Generated SDOF Properties there is a button that will plot out the static resistance function, this is a plot of the resistance function with out any of the dynamic effects. The actual dynamic resistance function will have a larger ultimate resistance.

Bilinear and Multi-step Resistance: These are similar resistance functions both start out elastic and end with a perfectly plastic resistance. The Multi-step has additional elastic plastic region. These resistance functions are based on yield line theory found in TM 5-1300 and UFC 3-340-1 and are representative of the flexure response of the member. The bilinear and the multi-step resistance deflection functions used in SPAn32 are determined following the procedures discussed in the DAHS CWE manual Chapter 10 beginning in Section 10.9 RESISTANCE FUNCTIONS.

Positive and negative phase regions: SPAn32 handles both positive and negative phases of the resistance function independently; they don't have to be mirrored images of the other. The positive and negative phases may differ; this happens in reinforced concrete members when the reinforcement is different in each face of the member.

Rebound hysteresis rules: SPAn32 uses the following rules to model rebound hysteresis in the resistance.

1. Rebound will always follow the elastic slope.

2. If stiffness slope is negative (softening stiffness) any rebound will cap the resistance at the resistance where rebound occurs. Rebound is elastic.

3. If stiffness slope is positive rebound will follow the elastic slope

4. Positive and negative phase resistance is handled independently. If rebound occurs from the positive phase resistance, the negative phase resistance will load normally accounting for the permanent offset that occurred in the positive phase.

Blast side, Other side: This is for the reinforced concrete member type. This identifies the face of the member with respect to the weapon location, or the direction the uniform load is coming from. Blast side is the front face or same side the blast load is coming from. Other side is the back face, or the face of the member opposite the blast load

Depth of Cover to Center: This is for the reinforced concrete member type, it is the depth of concrete from the nearest member face to the center of the reinforcing bars.

Type II cross-section: If the user has selected the Type II cross-section option in the reinforced concrete material input, SPAn32 will reduce the member moment capacity

when the member response exceeds 2 degrees rotation. This reduction in moment is to reflect the crushed concrete in the compression face being unable to carry compression. The moment capacity is changed to the value found in UFC 3-340-1 eq. 10-14.
Membrane Resistance

Dynamic Increase Factor (DIF): Materials in highly dynamic loading conditions exhibit increase strength as a function of strain rate. Increasing strain rate results in larger dynamic increase factor. This relationship allows increase in the material strength in the member.

Support Rotations: In the case of two-way members, the support rotation is determined as the average support rotation. When the distance from a support to a point of maximum deflection is not the same for all supports of a given member, the support rotation is calculated using the average of the distances.

Shear Design Summary: In the SPAn32 output equivalent support reactions used to calculate design shear stresses are based on the application of r_u as a unit static load. For two-way members a reduced support reaction is used near corners as illustrated in Figure 3-41 of TM 5-1300.

Model Types

SPAn32 has several options relating to modeling of the SDOF system. There are two principal material types, concrete and steel. There are two modes of structural behavior one-way and two-way. There is an option for a user defined bilinear resistance. These options will allow a large number of different building components to be modeled.

One-way Concrete Slab: This option will model any one way concrete member with a rectangular cross section. The section width for analysis is assumed to be a unit width (mm or inches). The loading of this member is in (psi or MPa) assuming the same unit width for the member. If the loaded area is different than the width of the member the load will need to be multiplied by a loaded area to member unit width factor. SPAn32 requires as a minimum one layer of reinforcing steel in either the blast face or the other face of the reinforced concrete member. This model is not limited to reinforced concrete only, it is able to model reinforced CMU by modifying the strength to account for the material and the cross-section properties.

Two-way Concrete Slab: This option will model any two-way reinforced concrete slab. SPAn32 requires as a minimum one layer of steel in the X-span and the Y-span directions making two layers of steel each orthogonal to the other in either the blast face or the other face of the reinforced concrete member.

One-way Steel Beam: This option will model any steel shape, standard AISC or custom. Inputs are provided for moments of inertia, elastic and plastic section modulus, these allow for the definition of any cross-section shape. In addition to the AISC shapes, open

web joist, corrugated steel decking and steel studs. These additional shapes would be limited to flexure response and their resistance limited to a bilinear form.

One-way Steel Plate: This option will model any one-way steel member with a rectangular cross section. The section width for analysis is assumed to be a unit width (mm or inches). The loading of this member is in (psi or MPa) assuming the same unit width for the member. If the loaded area is different than the width of the member the load will need to be multiplied by a loaded area to member unit width factor.

Two-way Steel Plate: This option will model any two-way steel plate.

User Defined: This option allows the user to define a generic bilinear resistance function and develop an equivalent SDOF model. The resistance function is mirrored for the negative phase; so both positive and negative are equivalent.

Model Type Options	**Concrete**		**Steel**			**User Defined**
	1-way	**2-way**	**1-way Beam**	**1-way Plate**	**2-way**	
Special Features						
Strain Hardening	X	X	X	X	X	
Automatic DIF calculation	X	X	X	X	X	
Deep Beam	X	X				
Concrete Tension DIF	X	X				
Membrane Resistance Tension &/or Compression	X	X				
Material Properties						
Concrete Compressive Str. f'_c	X	X				
Steel Yield Str. f_y	X	X	X	X	X	
Steel Ultmate Str. f_u	X	X	X	X	X	
Concrete Splitting Tension DIF	X	X				
Concrete Compressive DIF	X	X				
Steel Yield Strength DIF	X	X	X	X	X	
Steel Ultimate Str. DIF	X	X	X	X	X	
Type II cross section (conc. only)	X	X				
1 Way Support Conditions						
Fixed-fixed	X		X	X		
Simple-simple	X		X	X		
Fixed-simple	X		X	X		
Cantilever	X		X	X		
Fixed-Fixed Membrane	X					

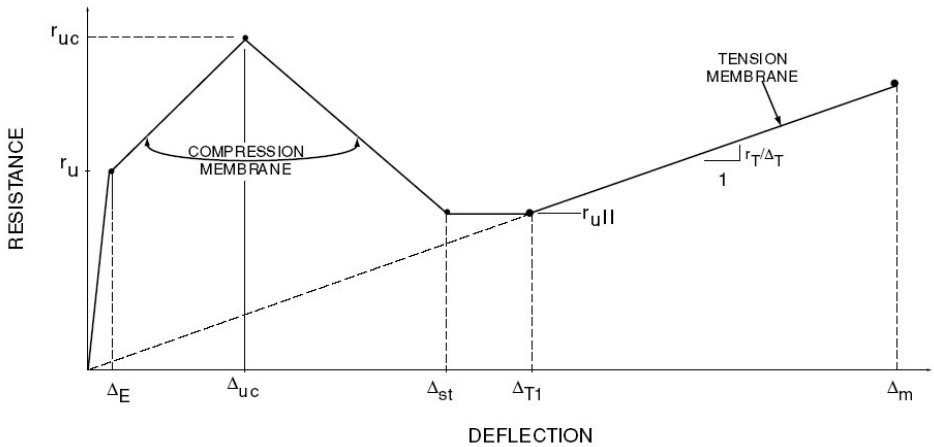
2 Way Support Conditions						
Fixed 4 sides		X			X	
Fixed 3 sides 1 free		X			X	
Fixed 2 adjacent sides 2 free		X			X	
Fixed 4 sides Membrane		X				
Simple 4 sides					X	
Simple 3 sides 1 free					X	

Table 1. Member Type, Support Conditions and Resistance Options

Special Features

Deep Member: Unrestrained deep members are defined as those with an aspect ratio length to thickness are for simply supported ≤ 2.0 and for continuous support ≤ 2.5. Restrained deep members are defined as those with an aspect ratio length to thickness L/h < 5. For laterally restrained Deep members compression only membrane behavior provides significant flexural strength. Deep members behave differently than normally proportioned members, lacking ductility, with special considerations for shear, and flexural response. SPAn32 checks member length to thickness ratio and applies the proper flexure and shear effects for deep members in accordance with UFC 3-340-1 Section 10.5.

Membrane Resistance: Membrane enhanced flexure resistance deflection functions are available only for 1-way and 2-way reinforced concrete with all supports fixed. This resistance function models the behavior of laterally restrained reinforced concrete in the membrane regime and are based on results of experimental research. The experimental stiffness of surrounds was quantitative and not based on strength. Though when used in design reinforcing details should follow the recommendations of UFC 3-340-1. The membrane-enhanced resistance deflection function used in SPAn32 follows the procedures discussed in the UFC 3-340-1 Chapter 10.4 CONCRETE MEMBRANE-ENHANCED FLEXURE and illustrated in Figure 1 below. SPAn32 options are for compression and tension membrane, tension membrane, and compression membrane only. The UFC 3-340-1 discusses three types on membrane regimes compression and tension, compression only (typically for restrained deep members) and tension only.



r_{uc} = COMPRESSIVE MEMBRANE ULTIMATE RESISTANCE TO UNIFORM LOAD (PARAGRAPH 10.4.2)

Δ_{uc} = CENTRAL DEFLECTION OCCURING AT r_{uc} (PARAGRAPH 10.4.2)

r_{uII} = FLEXURAL RESISTANCE USING TYPE II ULTIMATE MOMENT CAPACITY (PARAGRAPHS 10.3.3.2 AND 10.9.1)

r_T/Δ_T = TENSILE MEMBRANE RESISTANCE/CENTRAL DEFLECTION (PARAGRAPH 10.4.3)

Δ_m = MAXIMUM CENTRAL DEFLECTION, CORRESPONDING TO DESIGN RESPONSE LIMIT (PARAGRAPH 10.10)

Δ_{st} = CENTRAL DEFLECTION AT SNAP-THROUGH (PARAGRAPH 10.4.2)

Δ_{T1} = CENTRAL DEFLECTION WHERE TENSILE MEMBRANE SLOPE THROUGH ORIGIN REACHES r_{uII} (BUT $\Delta_{T1} \geq \Delta_{st}$)

r_u = ULTIMATE FLEXURAL RESISTANCE TO UNIFORM LOAD (PARAGRAPH 10.9.1)

Δ_E = EQUIVALENT ELASTIC DEFLECTION IN FLEXURE (PARAGRAPH 10.9.1)

Figure 1. Resistance Function for Membrane-Enhanced Flexure in Laterally Restrained Concrete Members. (UFC 3-340-1)

Dynamic increase factor

Both concrete and steel material strengths are strain rate dependent. The UFC-3-340-1 Chapter 4 Material Properties includes a lengthy discussion on the topic of strain rate behavior, section 4.5 discusses Cement-Based materials and section 4.6 discusses Steels. The material strength dynamic increase factors calculated by SPAn32 are based on the strain rate effect curves in UFC-3-340-1

Concrete compression DIF: This factor is used to increase the static concrete compressive stress because of increase strength seen in the strain rate effects in a dynamic system. The strain rates used are based on UFC 3-340-1 Figure 4-39. Design Curve for DIF for Ultimate Compressive Strengths of Concrete. For the concrete compressive strengths range 17.25 MPa < f'_c < 48.3 MPa. Acceptable range for the concrete compression DIF is 1.0 ≤ CDIF ≤ 1.9, if these values are exceeded a warning will issued in the text output.

Concrete Splitting Tension DIF: This factor is used to increase the static concrete splitting tension stress because of the increase strength seen in the strain rate effects in a dynamic system. The strain rates used are based on UFC 3-340-1 Figure 4-40. Design Curve for DIF for Concrete Splitting Tensile Strength. Acceptable range for the concrete splitting Tensile strength DIF is 1.0 ≤ CTDIF ≤ 70. If these values are exceeded a warning will issued in the text output. The effect of this factor on the member is to reduce the need for stirrup shear reinforcing.

Steel Yield Stress DIF: This factor is used to increase the static steel yield stress because of increase strength seen in strain rate effects a dynamic system. The strain rates used are based on UFC 3-340-1 Figure 4-48. Design Curves for DIF for Tensile Yield and Ultimate Stress of ASTM A615 Grades 40 and 60 Reinforcing Steel. Acceptable range for the steel yield stress DIF is $1.0 \leq SDIF \leq 1.75$. If these values are exceeded a warning will issued in the text output.

Steel Ultimate Stress DIF: This factor is used to increase the static steel ultimate stress because of increase strength seen in strain rate effects a dynamic system. The strain rates used are based on UFC 3-340-1 Figure 4-49. Design Curves for DIF for Tensile Yield and Ultimate Stress at Various Strain Rates for ASTM A36, A242 and A514 Steels. Acceptable range for the steel ultimate stress DIF is $1.0 \leq UDIF \leq 1.30$. If these values are exceeded a warning will issued in the text output.

Auto calculation for DIF: A powerful feature in SPAn32 is auto calculation of the dynamic increase factors. The strain rates used in auto calculation of dynamic increase factors for material strength are obtained by the following procedure. in SPAn32. The initial DIF values specified in the input data will be used as seed values for the initial DIF values prior to iteration

a) ***Ductility (μ)*** < 1.0, member remains elastic. The strain rate used is the average strain rate for the response period between the time of maximum resistance and the nearest previous minimum on the resistance history.

b) ***Ductility (μ)*** $\geq$ 1.0, plastic response. The strain rate used is the average strain rate during the period between the time of first yield and the nearest previous minimum on the resistance history.

The 'time to yield, equivalent' given in the SPAn32 output response summary is the amount of time required to go from zero strain to yield at the average strain rate as described in a) and b) above.

Strain Hardening: SPAn32 accounts for the effects of strain hardening in steel materials. This is a phenomenon that occurs when steel strain continues to increase after yield, the region where stress remains constant while strain increase is the plastic yielding region. In the higher strains the stress beyond the plastic yielding region begins to increase. This increase of strength of the steel is called strain hardening. This has the effect of increasing the dynamic steel stress f_{ds} this has the effect of increasing the ultimate resistance of the member.

Integration Method

There are two implicit integration methods used in the SPAn32 program, they are Constant Velocity and the Newmark Beta with integration constants γ=0.25 and β=0.5. The Constant Velocity method is used for time steps $\geq$ 0.01 ms and Newmark Beta for

time steps < 0.01 ms. Both methods are used because of inherent strengths and weakness of both. All SDOF calculations depend on integration routines. All integration routines exhibit time step sensitivity. One way that time step sensitivity manifests itself is in the peak response of the member. Most of the time the variations in the peak response will be small. However there are cases of instability where the variations of peak response will vary widely. The Constant Velocity method provides very good results for larger time steps, but exhibits instability when time steps are too small. Conversely Newmark Beta performs better for smaller time steps. One solution to these instabilities is to use only smooth load functions and limit the number of time steps allowed to some finite number. This solution would limit the types of problems that SPAn32 could be used for, while currently SPAn32 will handle the most complex transient loads generated from BlastX or other programs easily. Memory allocation is used in SPAn32 for the spatial arrays the only limitation in the time step and number of time steps used in an SDOF calculation is the availability of free memory on your computer. Not having a limit in size of spatial arrays can lead to very large number of time steps and very small time steps. It is important not to be limited by the size of the time step or the number of time steps.

SDOF Solution Process: There are multiple iterations performed in the SPAn32 SDOF solution depending on selection of several analysis enhancements. These enhancements are automatic dynamic increase factor calculation, strain hardening, and convergence of load mass factor and ductility. An attempt is made for each of these to converge on a stable answer. If convergence conditions were not satisfied a warning message is issued in the text output. If convergence conditions are successful there will be no warning messages.

SDOF Properties and Time Step

Before an SDOF calculation is performed a user is prompted to select a time step and duration. If the user selects analysis defaults, then calculated default time step and calculation duration will be used. Or the user can enter values for time step and calculation duration. In the "Generated SDOF Properties" dialog, information is presented to help the user select the proper time step and duration. This information is the natural period of the member, and the minimum time step and duration of the transient load file. The natural period of the member is the period of time it takes for the member to fully respond to the transient load. The time step based on the natural period should not be larger than 1/10 of the natural period. Typically a user will want to see a calculation duration that is more that 2-3 times the natural period. The minimum time step and the duration of the transient load should not be over looked. While it may not be feasible to use the smallest time step of the transient load the user needs to ensure that excessive clipping of load does not occur. The calculation duration should be greater than the duration of the transient load

The default time step and duration calculated by SPAn32 are based on some simple rules. These rules will work for most cases, but the user still needs to exercise care and judgement for the results of the calculation. The default time step is based on the least of 10 times the minimum transient load time step or 1/10 of the member natural period, this

is to arrive at a reasonable time step. The default calculation duration is based on the greatest of transient load duration or four times the natural period.

Equivalent System Mass

The system equivalent mass used in SDOF is calculated using a weighted-average of the elastic and plastic load mass factors based on the ductility ratio at peak deflection. The load mass factor used in SPAn32 is given by the eq. 1. SPAn32 uses the weighted-average of K_{LM} to conserve energy and momentum that would be lost when changing the system equivalent mass when passing from elastic resistance to perfectly plastic resistance.

$$K_{LM} := \frac{K_{LMe} + (\mu - 1) \cdot K_{LMp}}{\mu} \qquad \textbf{eq. 1}$$

where,

K_{LM} = weighted-average load-mass factor
K_{LMe} = elastic load-mass factor
K_{LMp} = plastic load-mass factor
μ = ductility ratio corresponding to the peak deflection.

Since ductility ratio is not known until after the analysis has been run, an iterative procedure is used until K_{LM} and μ converge. If convergence is not realized SPAn32 will issue a warning, see the warning discussion.

Load Definition

Currently no internal load calculations are performed. Dependence is on programs like CONWEP, BlastX, for airblast, GShock and FOIL for ground shock, or user supplied load-input files. Load data is acceptable in several different file formats, and several different unit combinations. Additional loading options are external static load, weight of an external mass acting with the member, and soil structure interaction (SSI). For an undamped SDOF system the external static load and attached mass effect the SDOF equation as shown below in eq 2.

$$\frac{\left[Resistance(x) - \left(P(t) + F_{static}\right)\right]}{\left(K_{LM} \cdot Mass_{member} + K_{LM} \cdot Mass_{attached}\right)} := a(t) \qquad \textbf{eq. 2}$$

Where,

resistance(x) =member resistance
P(t) =load history
F_{static} =external static load

$Mass_{member}$	=Member mass
KLM	=load mass factor
$Mass_{attached}$	=weight of attached external mass
a(t)	=acceleration

External Static Load: A static load applied to the structural member with positive being in the same direction as the blast load. This results in an initial static deflection before the pressure load is applied. It is entered as a unit load, e.g. psi. It is not attached physically to the member and does not increase the mass of the system. It may be an unattached equipment, or pooled liquid on the member. See eq.2

Weight of Attached External Mass: An attached external mass assumes that the external mass acts with the mass of the structural member during the dynamic calculations. It is entered as a unit load, e.g. psi. This is effectively additional mass to the mass of the system and will effect the natural period of the member. See eq. 2

Load Direction: The member being loaded has an orientation, and the load itself has a direction. The user selects the direction of the load from one of the three choices available horizontal, upward, or downward. The selected direction has an impact as to how the gravity load effects the member. If the load is horizontal then gravity has no effect. If the load is upward or downward then gravity plays a part in the dynamic loading. The direction of the blast load, if in the same direction as gravity the effect will be additive, if in opposite direction then the effect will be the difference of the two. This has the effect accounting for the weight of the member as a static load.

Soil Media Interaction: The soil structure interaction option is available for the modeling of members subjected to either active or passive ground shock loading; see Figure 2. Active ground shock loading occurs when the soil is the media that the ground shock travels through in loading the wall. This option requires entry of a value for acoustic impedance of the soil, ρc. Where ρ is the mass density of the soil and c is the shock front velocity (imparts a compressive stress in the soil). The value of ρc is used during the analysis to modify the load applied to the member at each time step according to the eq 3.

Interface stress for active ground shock loading.

$$\sigma_I := \sigma_{ff} + F_{Static} + \rho \cdot c \cdot \left(v_{ff} - v\right) \qquad \textbf{eq. 3}$$

Interface stress for passive ground shock loading

$$\sigma_I := \sigma_{ff} + F_{Static} + \rho \cdot c \cdot v \qquad \textbf{eq. 4}$$

where,

σ_I = interface stress, the load applied to the member

σ_{ff} = free field stress in the soil
F_{Static} = external dead load pressure (does not include member dead load)
ρc = acoustic impedance of soil
ρ = mass density of the soil
c = shock front velocity
v_{ff} = free field soil particle velocity
v = member velocity.

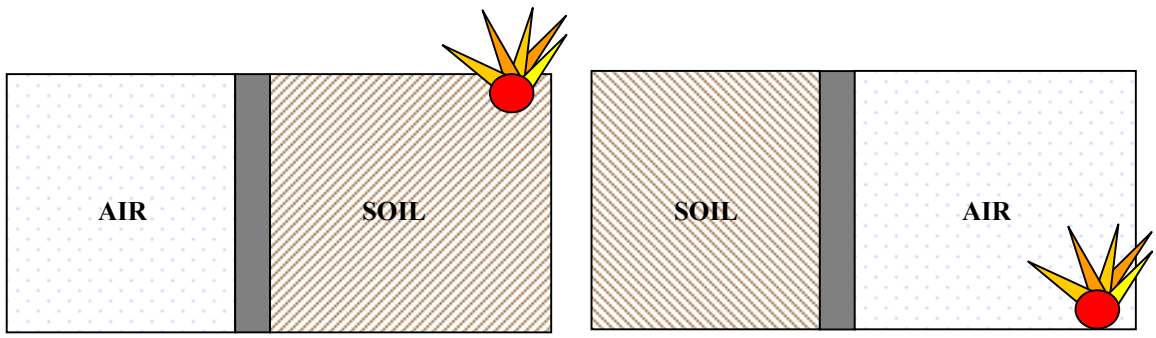


Figure 2. Active Ground Shock Loading **Passive Ground Shock Loading.**

Graphical Output

SPAn32 will plot a variety graphical output through the DPLOT95 program. These plots can be displayed in either SI or US units regardless of the original units of the data. The following plot types are available:

1. Input pressure load history
2. Input soil velocity history
3. Interface pressure @ supports
4. Interface Pressure @ max deflection point
5. Member resistance history
6. Member displacement
7. Member velocity
8. Member acceleration
9. Dynamic Reaction History
10. Dynamic Resistance vs displacement

Support Reaction histories: The support reaction histories output by SPAn32 are calculated based on the dynamic reaction equations given in tables 10-1 through 10-7 of UFC 3-340-1. It was necessary to extrapolate from these equations to obtain support reactions for all of the two-way member configurations available in SPAn32. The dynamic reaction equations are of the form

$$V_a(t) := C_1 \cdot R(t) + C_2 \cdot P(t) \qquad \textbf{eq. 5}$$

$$V_b(t) := C_3 \cdot R(t) + C_4 \cdot P(t) \qquad \textbf{eq. 6}$$

where,

V(t) = Shear history @ supports a and b
R(t) = member resistance at time, t
P(t) = load at time, t
C_1, C_2, C_3, C_4 are coefficients that determine the proportioning of reactions to account for both loading and inertial forces.

The support reaction history is used to develop the dynamic load for the supports of the member being analyzed. If you are analyzing a roof member, the dynamic reaction history would be used to design the supporting members for the roof member.

Warning Messages: SPAn32 makes a concerted effort to capture runtime warnings. The following is a list of warnings that may be issued in the text output displayed on the screen.

!!! AutoDIF calculation failed to converge!!! The DIF used in the calculation did not converge in the iteration process, rerunning the problem again may solve the problem. Also adjusting the time step larger or smaller may allow the problem to converge.

!!! Warning steel rebar exceeds 0.75 balanced design!!! Exceeding 0.75 balanced design my lead your concrete member to fail catastrophically. Staying within the bounds of a 0.75 balanced design according to ACI 318-89 requirements will allow the steel reinforcement to yield.

!!! Warning calculation time step is greater than 1/10-of the structural natural period. !!!The user was warned of this in the program input, the warning was a soft warning. The user is being reminded; the time step used the SDOF calculation may lead to inaccurate response. Using a time step less than 1/10-of the structural natural period will accurately capture the spatial response of the structural member in the SDOF calculation.

!!! A numerical integration instability has been detected exhibited by a constant velocity condition, !!! Examine your displacement plot, if the displacement has not reached its peak you may need to increase the calculation duration or if your calculation is taking off you may need to increase or decrease your time step. Currently SPAn32 uses a Constant Velocity integration for time step ≥ 0.01 and Newmark Beta λ=0.25 and β= 0.5 for time steps < 0.01.

!!! The steel strain rate is outside the bounds of the SDIF design curve; SDIF was capped !!! The user may have to adjust the time step for the SDOF calculation and rerun the problem. See UFC 3-340-1 Figure 4-48. Design Curves for DIF for Tensile Yield and Ultimate Stress of ASTM A615 Grades 40 and 60 Reinforcing Steel.

!!! The concrete compression strain rate is outside the bounds of the CDIF design curve; CDIF was capped !!! The user may have to adjust the time step for the SDOF calculation and rerun the problem. See UFC 3-340-1 Figure 4-39. Design Curve for DIF for Ultimate Compressive Strengths of Concrete.

!!! The concrete splitting tension strain rate is outside the bounds of the CTDIF design curve; CTDIF was capped !!! The user may have to adjust the time step for the SDOF calculation and rerun the problem. UFC 3-340-1 Figure 4-40. Design Curve for DIF for Concrete Splitting Tensile Strength.

!!! Excessive Member Rotation !!! This warning is set for rotation values > 30 degrees. The user needs to increase the capacity of the member being analyzed, or reduce the dynamic load.

!!! SPAn KLM failed to converge in Integration!!! The user will have to adjust the time step and rerun the calculation.

Comparative Analysis with DYNA3D

Objective: Verify that the SPAn32 SDOF modeling methodology is comparable to results obtained with DYNA3D

Background: SPAn32 is a computer program developed by the U.S. Army Corps of Engineers Protective Design Center (PDC) for the purpose of modeling the dynamic response of structural members subject to transient loading, such as explosive airblast. DYNA3D is an explicit finite element program developed by Lawrence Livermore National Lab, a U.S. Department of Energy Lab.

Load Definition: The blast loads used were positive phase only airblast predictions using CONWEP. Explosive weights used ranged from 100 kg to 10000 kg of TNT with a standoff distance of 60 meters. Thus providing a wide range of peak pressure and impulse values for the loading conditions. This range of airblast loads will allow member peak response to vary from elastic (with a ductility of about $\mu = 0.5$) to heavily plastic.

SPAn32 Model: A typical reinforced concrete slab member was modeled in SPAn32 for comparative analysis. In order to focus on the SPAn32 SDOF algorithm, a one-way reinforced concrete slab member was modeled, with all dynamic increase factors fixed at 1.0, and the effects of strain hardening turned off. The analysis was performed for two different reinforced concrete wall sections. One with a thickness of 300 mm, reinforcing ratio of $\rho = 0.3$ % and, a one-way simply supported span length of 4500 mm for a length/thickness ratio of 15:1. The other had the same thickness but with a reinforcing ratio of $\rho = 0.385$ % and a simply supported length of 6000 mm for a length to thickness ratio of 20:1. No gravity effects were accounted for in this analysis.

DYNA3D model: The model was created in DYNA3D using 8 node cube shaped brick elements with 4 elements through the slab thickness. The DYNA3D model was developed to match the SPAn32 resistance function as closely as possible. Matching the resistance function provided a control in the analysis and isolates the SPAn32 SDOF methodology for verification. The mass density of reinforced concrete used in DYNA3D was the same as that is used in SPAn32 (2304 kg/m^3). A linear elastic, perfectly plastic isotropic material model was used. Young's modulus was set to give the same member stiffness as was obtained with the SPAn32 model (i.e. the same initial slope of the resistance vs. deflection function). Yield stress was set to give the same ultimate resistance as was obtained with the SPAn32 model.

Results: There is a variation between the resistance vs. deflection function obtained with the DYNA3D model and the one obtained with SPAn32. This difference is shown in Figure 3. It was not possible to obtain identical resistance functions for the two models because of the plastic hinge formation process that occurs in the finite element model. The DYNA3D resistance function departs from its linear elastic slope to a reduced stiffness as yielding begins to occur. Once yielding at the plastic hinge location has progressed through the member thickness the plastic hinge is fully formed the DYNA3D model reaches the same ultimate resistance as SPAn32.

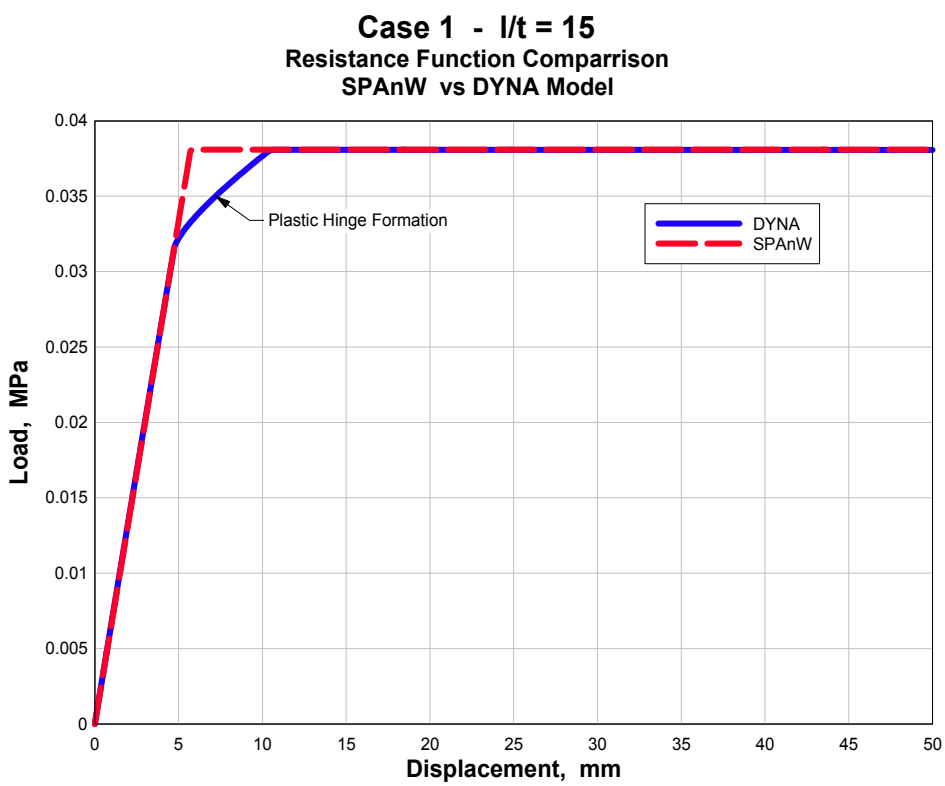


Figure 3. SPAn32 and DYNA3D Resistance Function.

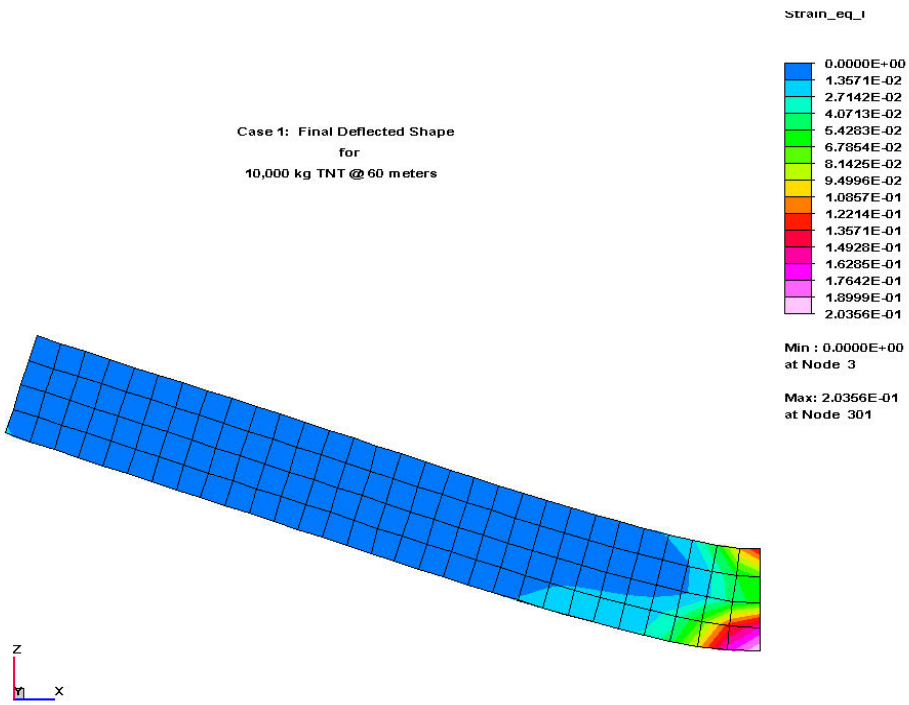


Figure 4. Final Deflected Shape Case 1 for 10000 kg TNT @ 60 m

The results of the comparative analysis show very good agreement between SPAn32 and the DYNA3D model. SPAn32 typically under-predicted the response calculated from DYNA3D. This can be partly attributed to the differences in the resistance functions. Closest agreement occurred when the ductility was less than one. It is also noted that beyond the elastic region agreement in peak response improves as the ductility increases. This trend may be occurring, as differences in the resistance function become less of a contribution to the peak response. Other differences may exist because of differences in the two methods.

Load, kg @ m	**SPAn32, mm**	**SPAn32**	**DYNA3D, mm**	**SPAn32/**
	Peak response	**Ductility**	**Peak response**	**DYNA3D ratio**
100@60	2.86	0.49	2.85	1.004
200@60	4.36	0.75	4.33	1.007
500@60	8.10	1.40	8.66	0.935
1000@60	17.73	3.06	19.16	0.925
2000@60	50.09	8.63	52.93	0.946
5000@60	217.6	37.5	221.5	0.982
10000@60	653.7	112.7	652.1	1.002
SPAn32 time step = 0.1 ms l/t = 15:1, thickness 300 mm, unsupported length = 4500mm, ρ = 0.3 %, elastic limit = 5.8 mm				

Table 2. Comparative Analysis Case 1.

Load, kg @ m	SPAn32, mm	SPAn32	DYNA3D, mm	SPAn32/
	Peak response	Ductility	Peak response	DYNA3D ratio
100@60	5.63	0.46	5.67	0.993
200@60	8.89	0.72	8.95	0.993
500@60	17.09	1.39	18.26	0.936
1000@60	35.72	2.91	38.14	0.936
2000@60	90.14	7.34	95.47	0.944
5000@60	348.1	28.4	363.9	0.957
10000@60	991.0	80.7	1013.9	0.977
SPAn32 time step = 0.1 ms l/t = 20:1, thickness 300 mm, unsupported length = 6000mm, ρ = 0.385 %, elastic limit = 12.275 mm				

Table 3. Comparative Analysis Case 1.

Summary and Conclusions

SPAn32 as its name implies, is primarily a SDOF program for plastic analysis. Since this is the primary response of structural members in a blast environment, SPAn32 is a useful program for analysis of structural members that undergo significant deformation. The special features in the program attempt to model the dynamic response of the structural member by taking advantage of strain rate effects, membrane effects, and dynamic strength increase factors. SPAn32 program is a powerful SDOF tool for analysis and design not only of hardened facilities but also of conventional construction.

The SPAn32 SDOF methodology showed very good correlation with the DYNA3D model. Further testing and validation of the SPAn32 program are needed, specifically validation with test data.

There are several enhancements that will be made to SPAn32 in the future these include, 1) 12 steel model for reinforced concrete to provide more accurate support conditions, and reinforcement layout, 2) openings in a slabs and the resulting yield lines, A user defined multi-step resistance function, 3) improved numerical analysis, and 4) generation of Pressure-Impulse curves for given response.

REFERENCES

Structures to Resist the Effects of Accidental Explosions, ADA 243 272/TM 5-1300/NAVFAC P-397/AFR 88-22, Departments of the Army, The Navy, and the Air Force, November 1990.

Fundamentals of Protective Design for Conventional Weapons, Technical Manual No. 5-855-1, Headquarters Department of the Army, Washington, D.C., 3 November 1986.

Design and Analysis of Hardened Structures to Conventional Weapons Effects, DSWA DAHSCWE-97, Army TM 5-855-1, Air Force AF JMAN 32-1055, Navy NAVFAC P-1080, April 1997.

Biggs, J.M., Introduction to Structural Dynamics, McGraw-Hill Book Company, 1964

Park, R. and Gamble, W.L., Reinforced Concrete Slabs, Wiley Interscience, New York, 1980

Drake, J.L., et. al., Protective Construction Design Manual, prepared for Air Force Engineering & Services Laboratory, Tyndall Air Force Base, Florida, November, 1989.

A manual for the Prediction of Blast and Fragment Loadings on Structures, DOE/TIC–11268, United States Department of Energy, Albuquerque Operations Office, July 1992.

Hyde, D.W., ConWep Computer Application for US Army TM 5-855-1 Conventional Weapons Effects, US Army Engineer Waterways Experiment Station, Structures Laboratory, September 4, 2003.

Design of Structures to Resist the Effects of Atomic Weapons, Principles of Dynamic Analysis and Design TM 5-856-3, Department of the Army, U.S. Army Corps of Engineers, Washington, DC 20314-1000 (March 1957)

Design of Structures to Resist the Effects of Atomic Weapons, Structural Elements Subjected to Dynamic Loads TM 5-856-4, Department of the Army, U.S. Army Corps of Engineers, Washington, DC 20314-1000 (March 1957)

Hyde, D.W., DPLOT95 Computer Application for Protective Structures Automated Design System (PSADS), US Army Engineer Waterways Experiment Station, Structures Laboratory, September 4, 2003

Britt, R. J., BLASTX Computer Application for Protective Structures Automated Design System (PSADS)

Hyde, D.W., FOIL Computer Application for Protective Structures Automated Design System (PSADS)

Walton, B.A., GSHOCK – Ground Shock Computer Application for Protective Structures Automated Design System (PSADS)

Walton, B. A., Knight, T.C., Thomas, S.W., Automated Designer Computer Application for Protective Structures Automated Design System (PSADS)

DYNA3D, A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics, United States Department of Energy, Lawrence Livermore National Laboratory, 1996

Paper by Seipel, Thomas, Knight