

Accuracy of the Rubicon Toolbox Finite Element Model

Introduction

This document deals with the accuracy and recommended use of the Rubicon Toolbox Finite Element module. The document is intended to provide guidance to users about the strengths and limitations of the model, and to provide general guidance on how the Finite Element model can best be used to analyze pavement designs.

The Rubicon Toolbox Finite Element Model

The Finite Element (FE) tool allows researchers and pavement designers to perform a more in-depth and sophisticated analysis of stresses and strains within a pavement structure. The Rubicon Toolbox FE Tool incorporates an Axi-symmetric Finite Element model with a stress-dependent material model which allows you to analyze and view the likely stiffnesses that will occur at different locations within the pavement. The Rubicon Toolbox FE Model incorporates a mesh-generation function which automatically generates a mesh to provide optimal accuracy of stresses and strains. This means that you can simply point the tool to a predefined pavement structure, set the load and run the model.

The finite element tool thus strikes a convenient balance between sophistication and ease of use, and is an ideal tool to use for educational purposes or when trying to gain a more in-depth understanding of stress-strain patterns within pavement structures.

How to Use the Finite Element Tool

Because of the relatively sophisticated nature of the underlying Finite Element model, there are few transfer functions that have been derived for use in conjunction with stress-dependent stress and strain distributions. Also, because of the stress-sensitive material model, stresses and strains cannot simply be superimposed for more than one load, and the model therefore allows only a single applied load area. For these reasons, it is strongly recommended that the results obtained with the Finite Element tool be used in a relative and comparative manner, as opposed to the traditional Mechanistic-Empirical approach which seeks to obtain an absolute estimate of the number of repetitions to failure.

For relative and comparative analyses, the Rubicon Toolbox Finite Element tool provides the following functions:

- **Layer Sensitive Contour Plot** to view the distribution of a stress or strain component within a pavement structure. The contour plot shows where maximum stresses or strains

occur within layers and can be used to compare the stress distributions in different pavements in a relative manner.

- **Stiffness and Deformation Plot** to view the stress sensitive stiffnesses that are likely to occur at different locations within the pavement structure under loading.
- **Results Grid** function that generates a grid of stress and/or strain results at different coordinate points. Results can then be exported to Excel for further analysis of key patterns and trends at different depths or offsets from the load centre.

Stress-Sensitive Material Model

The finite element model can analyze isotropic linear elastic or isotropic non-linear (stress-sensitive materials) or a combination of these material types. The material model adopted for each layer will depend on the material properties set up for the pre-defined pavement structure, and specifically on the properties defined on the Stress-Dependency tab of the Materials Definition Window. For non-linear, stress-dependent materials the Rubicon Toolbox Finite Element model uses the well known Universal Soil Model (Uzan, 1992), which overcomes some of the deficiencies of the simpler and well known K1-Theta-K2 model.

The Universal Soil Model provides a single model with which fine and course grained materials can be modelled. The Universal Soil model has the following form (Uzan, 1995, Witczak and Uzan, 1998):

$$Stiffness = (K_1 \cdot Pa) \left(\frac{\Theta}{Pa} \right)^{K_2} \left(\frac{\tau_{oct}}{Pa} \right)^{K_3}$$

- Where:
- Θ = Bulk stress (sum of three normal or principal stresses).
 - K_1, K_2, K_3 = Material constants determined from laboratory tests.
 - τ_{oct} = Octahedral Shear Stress.
 - Pa = Atmospheric pressure, in the unit of stress (typically kPa or psi. Atmospheric pressure = 101 kPa).

Where:

$$\tau_{oct} = \frac{1}{3} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$$

- Where: $\sigma_1, \sigma_2, \sigma_3$ = Three principal stresses

The Octahedral Shear stress is a stress invariant (i.e. its value is not dependent on the orientation of the defined stress state) which provides a compact description of the overall distortion or shear aspect of any imposed stress state. In pavement systems, the octahedral shear stress in a layer would tend to be high under the following conditions:

- In a base or subbase that is poorly supported (higher bending at a given applied load);
- At the edges of the applied load;
- At the edges of cracks or discontinuities, and
- At the bottom of stiff layers which exhibit bending (e.g. cement-stabilized subbases).

The atmospheric pressure is used to provide the equation in manner such that K_1 , K_2 and K_3 are dimensionless as long as the stress components (Bulk stress and Octahedral shear stress) are in the same unit as the Atmospheric Pressure. Typical values for the parameters K_1 to K_3 in the universal soil model are shown in Table 1.

Table 1: Typical Values for the K_1 to K_3 in the Universal Soil model (after Jooste and Fernando, 1995)

Material	K_1 Range	K_2 Range	K_3 Range
Asphalt	30 000 to 60 000	0.0 to 0.2	0.0 to -0.3
Crushed Stone	500 to 1500	0.5 to 0.65	0.0 to -0.3
Sand-Subgrade	150 to 250	0.45 to 0.8	-0.3 to -0.9
Clay Subgrade	150 to 600	0.0 to 0.0	-0.1 to -0.3

When a material is defined as one that has a stress-sensitive stiffness in the materials definition window, the Finite Element model will iteratively calculate stresses and strains throughout the mesh. After each iteration the calculated stresses at the centre of each element in the mesh are used in the Universal Soil model to calculate an updated stiffness for each element. This process is performed in an iterative manner until the stiffness of each element has converged to within 5% or 5 MPa of the stiffness of the same element in the preceding iteration.

If convergence has not been achieved after 20 iterations the process stops automatically and the results are displayed. This seldom happens, but is needed to ensure the calculation does not run indefinitely because of failure to converge in one or two elements within the mesh. It should be noted that, since the stresses at the centre of each element are used to determine the stress-sensitive stiffness for the element, the stiffness of the element will not correspond exactly to the stress state at all locations within the element. This is an unavoidable aspect of this approach.

Aspects Related to Model Accuracy

It should be noted that the FE model, like all models, provides an approximation of the stress state within the pavement. Although the Rubicon Toolbox FE model is a tried and tested, and very accurate model, it is still an approximate model and complete agreement with exact theoretical models should not be expected at all locations within the mesh. This applies specifically at locations such as the pavement surface right at the edge of the load area. It should also be noted that the mesh dimensions are such that the subgrade thickness is effectively 5000 mm (5 metres), and thus complete agreement should not be expected between the Finite Element tool displacements and the displacements calculated using Layered Elastic Theory, where a semi-infinite subgrade is assumed.

To provide an indication of the accuracy of the Rubicon Toolbox Finite Element model, the following comparisons are provided:

Benchmarking against Strains Noted by Huang

In this analysis, some of the strains calculated by Huang (Huang 1993) in his analysis of several response models are compared against the values obtained using the Rubicon Toolbox Finite Element model for the case where all layers are linear elastic. The pavement used by Huang (see pages 149 and 150 of his text) is shown in Table 2.

Table 2: Pavement Structure used by Huang (1993) for Benchmark Analysis

Layer	Stiffness (in Mpa and [ksi])	Thickness (mm and [inches])	Poisson's Ratio
Surfacing	3450 [500]	102 [4]	0.30
Base	173 [25]	203 [8]	0.35
Subbase	104 [15]	203 [8]	0.35
Subgrade	34.5 [5]	Semi-infinite	0.35

In Huang, this pavement was evaluated under a 40 kN single load and for the following variations (Huang, 1993):

- ◆ Contact pressure of 483 kPa;
- ◆ Contact pressure of 966 kPa;
- ◆ Increase in subgrade stiffness from 34.5 to 103.5 MPa.

For each of these variations, stresses and strains were evaluated at various depths at the load centerline. Tables 3, 4 and 5 show the results as shown by Huang for the ELSYM5 layered elastic model, and as calculated using the Rubicon Toolbox Finite Element tool.

It can be seen from Tables 3 to 5 that the stresses and strains calculated with the Rubicon Toolbox Finite Element model generally compare very well with the ELSYM5 results reported by Huang. The error is generally less than 5%, except in some instances (highlighted in Tables 3 to 5) where the percentage error is higher mainly because of the low stress values used in the denominator of the percentage error calculation.

Further Benchmarking of Stress-Strain Trends against Linear Elastic Theory

A further analysis of the stress and strain trends was performed for the structure shown in Figure 1. For this structure, the stresses and strains were calculated with the Stress-Strain Calculator Tool of Rubicon Toolbox, which implements the WESLEA linear elastic engine. Stresses and strains were then also calculated using the Rubicon Toolbox Finite Element model. For both models, evaluation positions were chosen at the top, middle and bottom of each layer (except for the subgrade, which was evaluated only at the top of the layer). These evaluations were performed at the load centerline and just outside the edge of the load area.

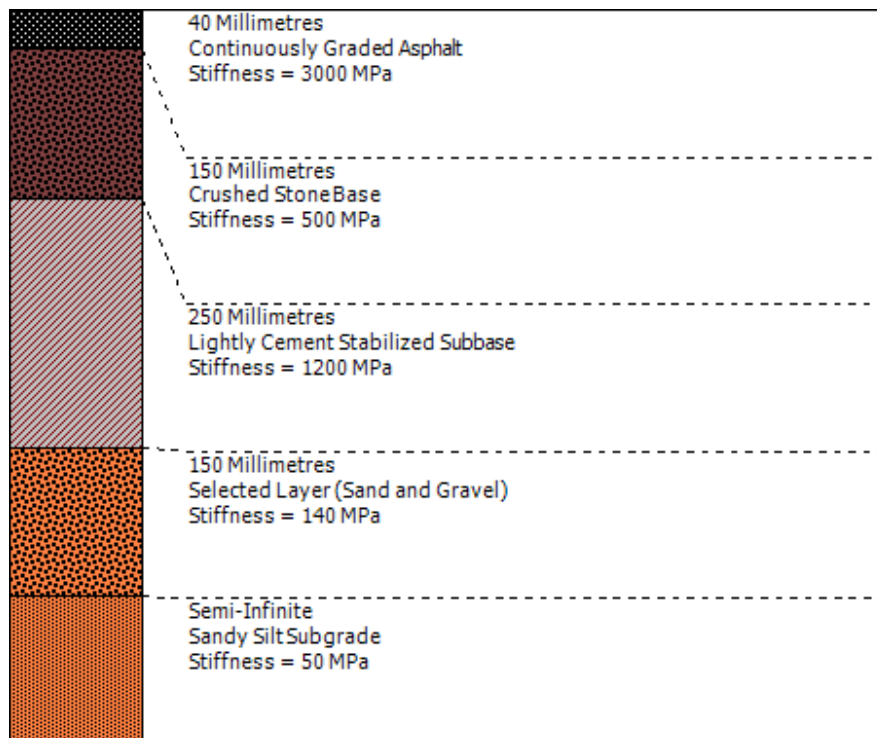


Figure 1: Pavement Structure Used in Stress-Strain Trend Comparison

Table 3: Comparison between Elsym5 and Rubicon Finite Element for Linear Elastic Case (Results after Huang (1993), for Applied Load of 483 kPa)

Position	Component	Elsym5 (from Huang)	Rubicon Toolbox FE*	% Error
Top of Surfacing	Vertical Displacement, in microns	864.0	769.0	12%
Bottom of Surfacing	Horizontal Stress, in kPa	-1379.7	-1363.0	1%
Bottom of Surfacing	Horizontal Strain, in microstrain	-295.7	-291.0	2%
Top of Base	Vertical Stress, in kPa	182.7	179.0	2%
Top of Base	Horizontal Stress, in kPa	19.9	17.2	< 5 kPa Diff
Bottom of Base	Horizontal Stress, in kPa	-48.0	-47.1	2%
Top of Subbase	Vertical Stress, in kPa	59.2	56.7	4%
Top of Subbase	Horizontal Stress, in kPa	-16.0	-15.5	3%
Bottom of Subbase	Horizontal Stress, in kPa	-36.2	-33.4	8%
Top of Subgrade	Vertical Stress, in kPa	24.0	23.8	1%
Top of Subgrade	Horizontal Stress, in kPa	2.7	-2.5	< 5 kPa Diff
Top of Subgrade	Vertical Strain, in microstrain	688.8	739.0	7%
			Average Error	4%

Note: FE model assumes a 5 m deep subgrade, while ELSYM5 assumes semi-infinite subgrade

Table 4: Comparison between Elsym5 and Rubicon Finite Element for Linear Elastic Case (Results after Huang (1993), for Applied Load of 966 kPa)

Position	Component	Elsym5 (from Huang)	Rubicon Toolbox FE*	% Error
Top of Surfacing	Vertical Displacement, in microns	914.0	809.0	13%
Bottom of Surfacing	Horizontal Stress, in kPa	-2010.6	-1991.0	1%
Bottom of Surfacing	Horizontal Strain, in microstrain	-428.7	-422.0	2%
Top of Base	Vertical Stress, in kPa	238.1	231.0	3%
Top of Base	Horizontal Stress, in kPa	14.3	9.6	< 5 kPa Diff
Bottom of Base	Horizontal Stress, in kPa	-52.2	-50.5	3%
Top of Subbase	Vertical Stress, in kPa	64.2	60.9	5%
Top of Subbase	Horizontal Stress, in kPa	-17.5	-16.6	5%
Bottom of Subbase	Horizontal Stress, in kPa	-37.7	-34.1	11%
Top of Subgrade	Vertical Stress, in kPa	24.7	24.2	2%
Top of Subgrade	Horizontal Stress, in kPa	0.1	-2.6	< 5 kPa Diff
Top of Subgrade	Vertical Strain, in microstrain	713.8	753.0	5%
			Average Error	5%

Note: FE model assumes a 5 m deep subgrade, while ELSYM5 assumes semi-infinite subgrade

Table 5: Comparison between Elsym5 and Rubicon Finite Element for Linear Elastic Case (Results after Huang (1993), for Stiffer Subgrade (103.5 MPa))

Position	Component	Elsym5 (from Huang)	Rubicon Toolbox FE	% Error
Top of Surfacing	Vertical Displacement, in microns	538.0	494.0*	9%
Bottom of Surfacing	Horizontal Stress, in kPa	-1316.5	-1304.0	1%
Bottom of Surfacing	Horizontal Strain, in microstrain	-283.5	-280.0	1%
Top of Base	Vertical Stress, in kPa	188.2	185.0	2%
Top of Base	Horizontal Stress, in kPa	26.2	23.4	< 5 kPa Diff
Bottom of Base	Horizontal Stress, in kPa	-29.6	-29.6	0%
Top of Subbase	Vertical Stress, in kPa	73.3	70.9	3%
Top of Subbase	Horizontal Stress, in kPa	-2.0	-1.9	< 5 kPa Diff
Bottom of Subbase	Horizontal Stress, in kPa	-3.8	-1.6	< 5 kPa Diff
Top of Subgrade	Vertical Stress, in kPa	39.1	38.5	1%
Top of Subgrade	Horizontal Stress, in kPa	24.5	-1.6	Large Error
Top of Subgrade	Vertical Strain, in microstrain	355.2	381.0	7%
			Average Error	3%

Note: FE model assumes a 5 m deep subgrade, while ELSYM5 assumes semi-infinite subgrade

A comparison of the stresses and strains calculated with the WESLEA and Finite Element models is shown in Figures 2 and 3, respectively. The applied load in both instances was a single 40 kN load with a contact pressure of 750 kPa.

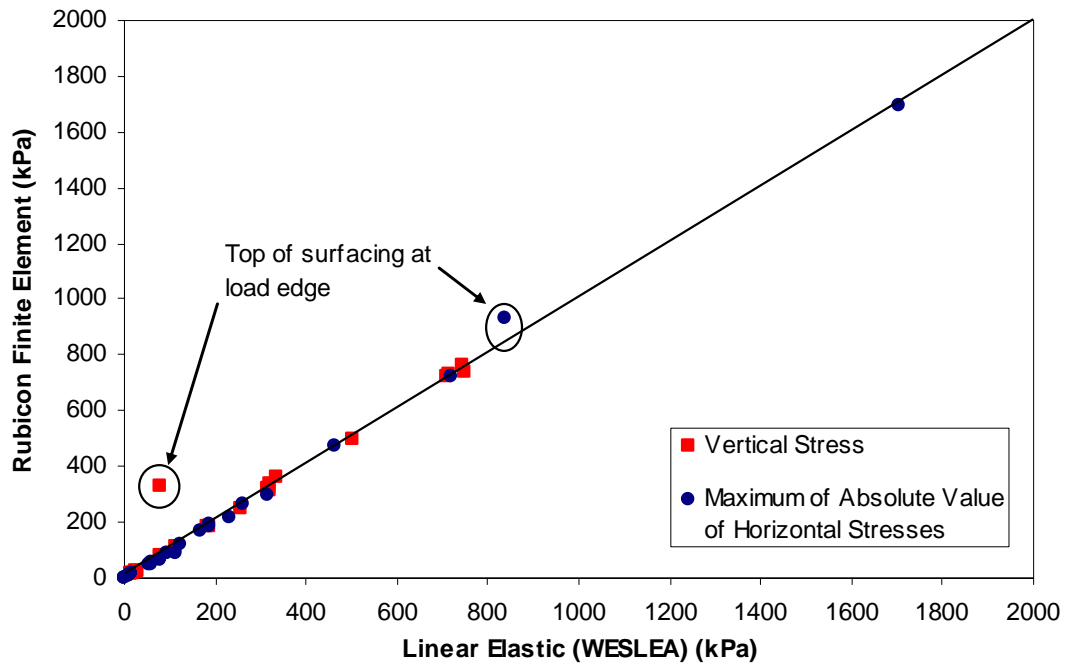


Figure 2: Comparison of Stresses calculated with Rubicon FE model and WESLEA

Figures 2 and 3 show that the absolute values of the stresses calculated by the WESLEA and the Rubicon Toolbox Finite Element modules compare well. Differences between the results calculated by the two models are very small when compared to the absolute values of the stresses and strains at each evaluation position. As highlighted in Figure 2, there are some exceptions, notably at the top of the surfacing and right at the edge of the load. However, at this location it can be expected that different models will yield different results, depending on the exactness of the integration techniques used in the model in this area which has a steep stress-gradient.

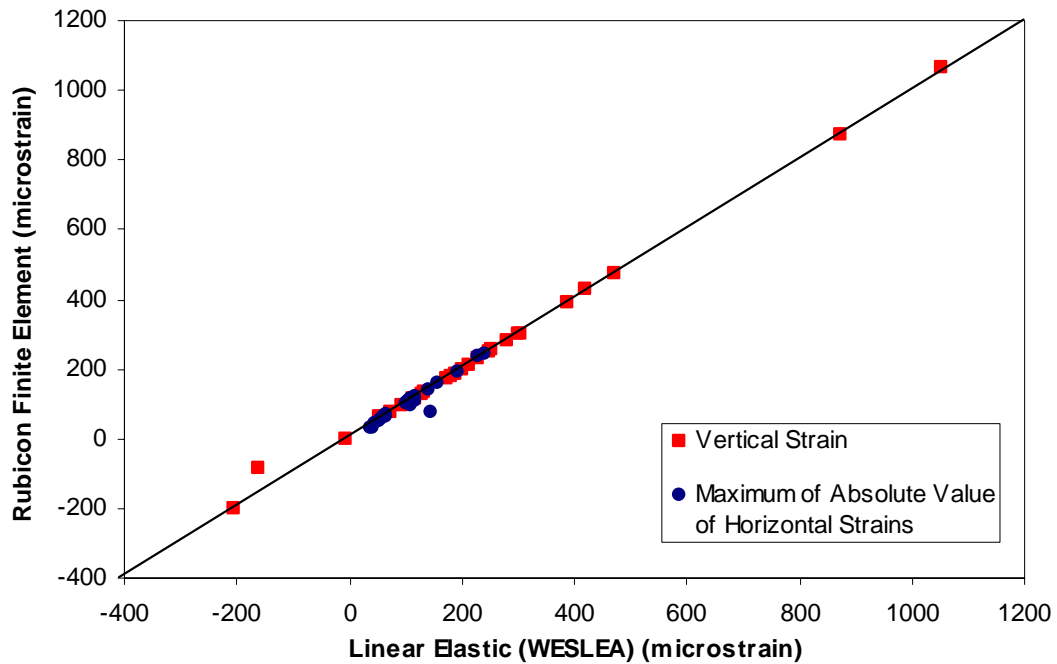


Figure 3: Comparison of Stresses calculated with Rubicon FE model and WESLEA

Calculation of Stress-Dependent (Non-Linear) Moduli

It should be noted that, in the Rubicon Toolbox Finite Element model, the stress-dependent model iteration uses the centre of each element to determine the stress-dependent stiffness. It should therefore not be expected that the stress-state will show complete agreement with the stress-sensitive stiffnesses at all locations within the mesh. However, the model provides a general indication of the likely stiffness at each location within the pavement, and is therefore a vast improvement in the layered linear elastic formulation which assumes a constant stiffness throughout the layer in the vertical and horizontal directions.

References

JOOSTE, F.J. and Fernando, E.G. 1995. **Development of a Procedure for the Structural Evaluation of Superheavy Load Routes**. Texas Transportation Institute: The Texas A&M University System. College Station, Texas. (Research Report 1335-3F).

UZAN, J. 1992. Resilient Characterization of Pavement Materials. **International Journal for Numerical and Analytical Methods in Geomechanics**. Volume 16, pp. 435 to 459.

UZAN, J. 1985. **Granular Material Characterization**. In Transportation Research Record 1022. TRB, National Research Council, Washington, D.C., 1985, pp 52 to 59.

WITCZAK, M.W. and Uzan, J. 1988. **The Universal Pavement Airport Design System, Report I of IV: Granular Material Characterization**, University of Maryland, College Park, MD, September 1988.