

UC-win/FRAME(3D) 5.01.00 International Version Updated Information

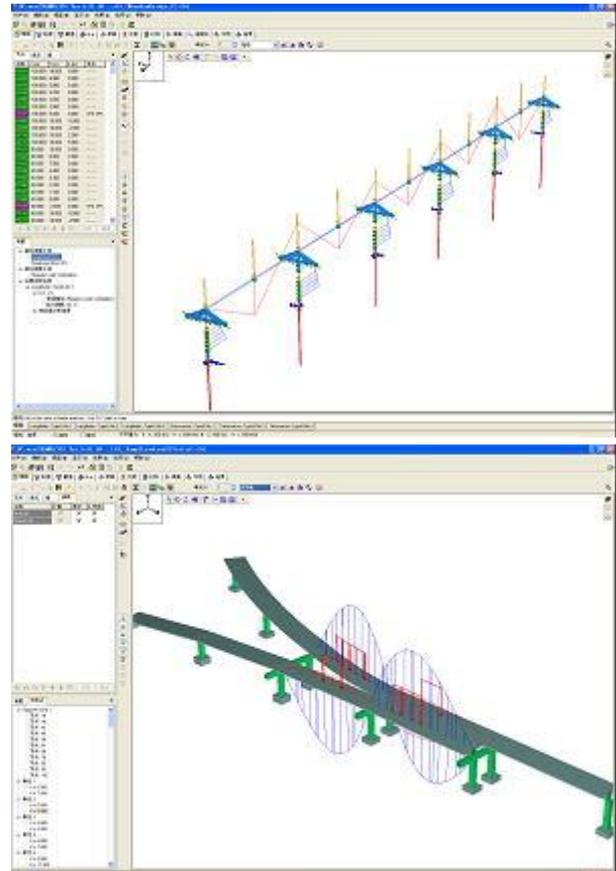
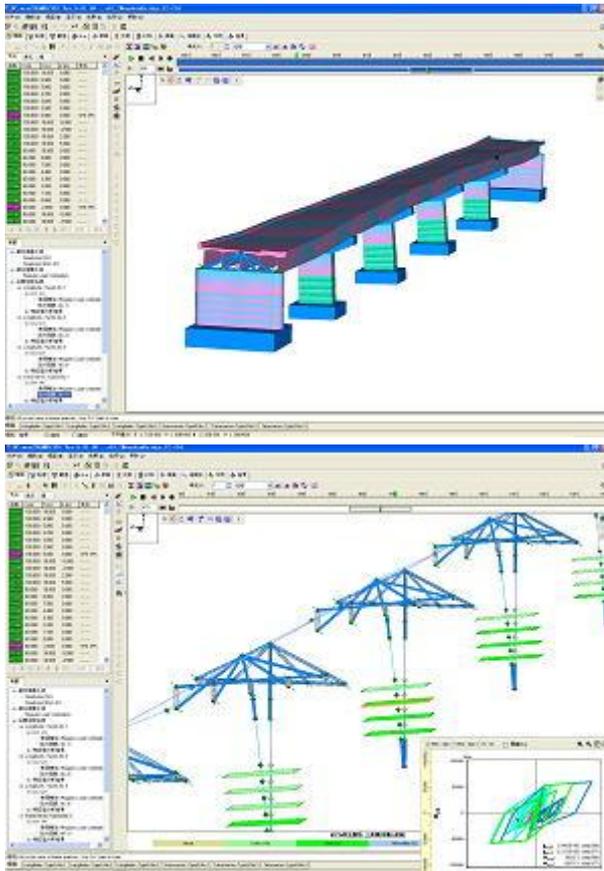


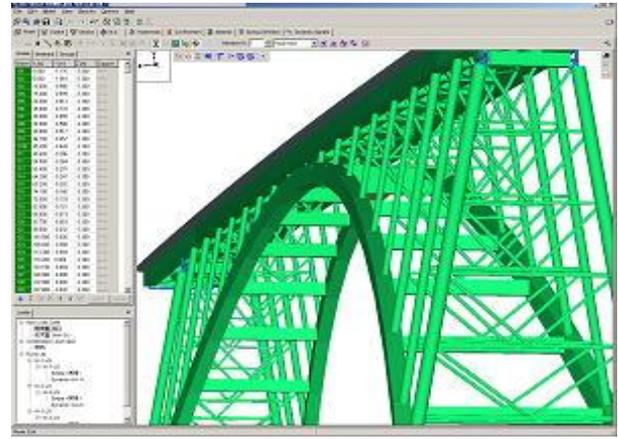
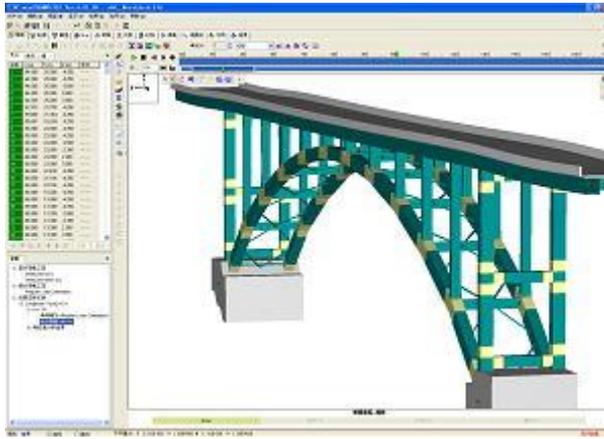
Thanks to the release of UC-win/FRAME (3D) 5.01.00 International Version by FORUM8, it is now possible to input and output in English and Chinese. Help files in English and Chinese have also been added.

Introduction

[UC-win/FRAME \(3D\)](#) is a general-purpose 3D frame analysis program. The Advanced version, a top-of-the-line software allows static and dynamic analysis considering both the material nonlinearity and geometry nonlinearity. It has the capability of verifying many aspects of beam structures including allowable stress, bending strength, shear strength, and nonlinear member.

In this revision of the software, the following three functions are added.





Reduced stiffness in the eigenvalue analysis (Automatic calculation of yield rigidity)

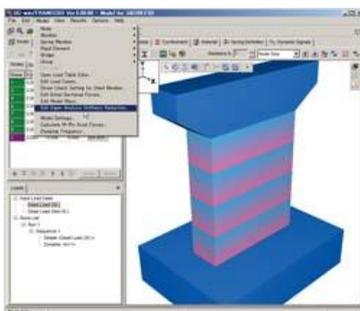
The following sentences are stated in the expository writing of "6.2.3 Calculation method of eigen cycle", "March 2002 Highway Bridge Specification V Seismic Design Volume, dynamic verification with wide application range, Japan road Association".

As for the stiffness of bridge pier rigidity of -----(snip)----- is used to verify if the bridge pier has the capacity to withstand level1 earthquake, and the yield rigidity of bridge pier is used to verify if it has the capacity to withstand level2 earthquake.

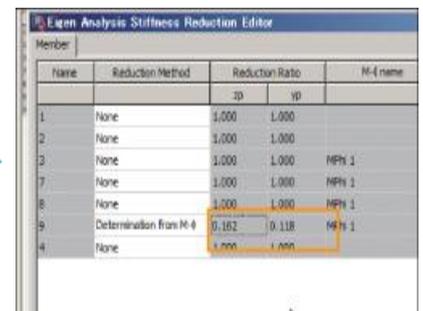
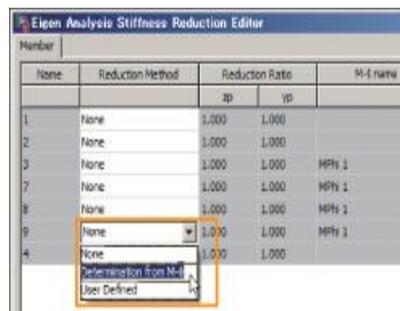
The whole bridge is designed with seismic capacity sufficient for absorbing energy when members become like plastic during a strong earthquake such as the level 2 one. When members become like plastic, their stiffness is reduced and the eigen cycle becomes long. Though it is necessary to calculate repeatedly based on the expository writing of the Highway Bridge Specification to do an analysis properly, an eigen cycle based on the yield rigidity as a matter of convenience.

In the previous version of FRAME(3D), the yield rigidity was automatically calculated from the section shape and axial force acting on members by setting property of M- to "bilinear" or "linear", in order to calculate the eigen cycle. When eigenvalue analysis is performed after selecting "Tri-linear" or "Tetra-linear" under the category of M- property, the eigen cycle based on the cracked stiffness is calculated. Therefore, when the time history response analysis is performed with "Tri-linear" or "Tetra-linear", a separate model was required for the eigen cycle calculation. In this revision of the software, a switch that can be turned on in order to apply yield rigidity to eigenvalue analysis even in such a case has been implemented. It is no longer necessary to calculate the yield rigidity separately.

1. Select "Edit the stiffness reduction value in eigenvalue analysis" from model menu (Figure1).
2. Select the element in question and choose the stiffness reduction method. User has a choice of selecting either the method where the ratio of cracked stiffness to yield rigidity is automatically calculated from M- ϕ property and applied as reduction ratio, or other method when it comes to reducing stiffness. (Figure2).

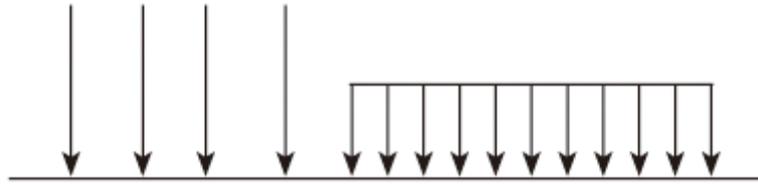


▲Figure1 Selection of model menu



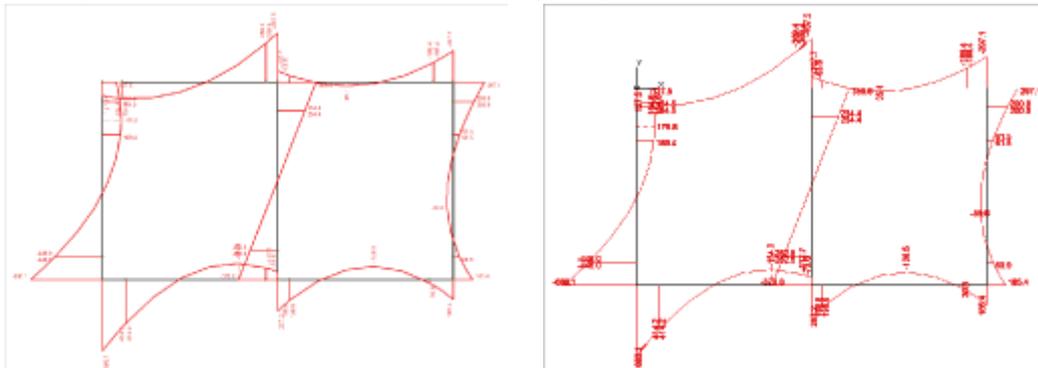
▲Figure2 Selection of the element in question and the method of stiffness reduction

UC-win/F-RAME(3D) has influencing line analysis function but the loading type was mainly based on the Highway Bridge Specification before this new version has been released. In this new version, EA load based on "April 2004 Railway Structure Design Specification, Concrete Structure" can now be input. The EA load is the combination of axle load group (multiple concentrated load) of the tracked train and the load of goods train behind the tracked train.



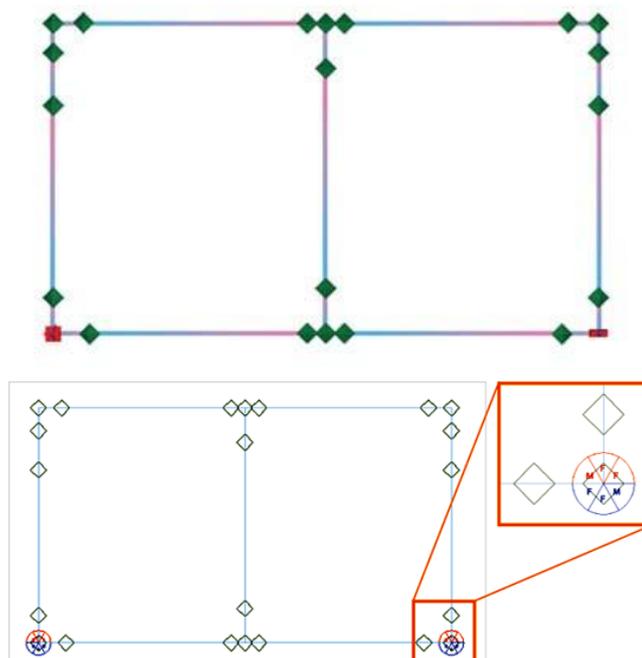
▲Figure3 Image of EA load input

Model and section force graph that has been illustrated after calculation can now be output to a file in DWG/DXF format. Therefore, they can be edited with CAD software (Figure4).



▲Figure4 Moment (Left: UC-win/F-RAME (3D), Right:DWG file)

In the analysis model, the fulcrum conditions are displayed as well as the elements and nodes(Figure5).



▲Figure5 Model diagram (Top: UC-win/F-RAME (3D), Bottom: DWG file)