INTRODUCTION
Visual Bridge Design System (VBDS) is a new generational bridge analysis and design software based on years’ experience on both bridge engineering practice and software development. Its design and development adopted entirely the state-of-the-art software development technology, such as relational database management, graphical user interface, visualization and object oriented graphic integration (Object ARX). The platform for the system is PC with Windows 2000/95/98/NT.

The core of the system consists of **FEA Kernel, Preprocessor, Postprocessor, and Prestress Tendon Configuration.** Through a seamless integration technology, the four subsystems are formed into one integrated system. The outstanding feature of the system is to adopt the unified core database. All four systems were programmed using the Microsoft Visual C++. These four subsystem are described as follows:

**FEA Kernel** is to conduct bridge structural analysis. It is specifically designed to meet the special requirements of the spatial structural analysis of a bridge built in multiple construction stages. It is purely developed with the object-oriented technology. Its main features are:

- Excellent graphic interface and visualization
- 3D beam, truss and shell elements
- Interchange between multiple construction stages
- Analysis of multiple load cases
- Specifically designed, bridge construction oriented loads
- Automatic adjustment for cable-stayed bridge internal forces
- Construction control analysis
- Prestressing force calculation
- Concrete creep and shrinkage analysis
- Influence surface and extreme live load analysis
- Dynamic modal analysis
- Elastic stability analysis
- Large deformation and other geometry non-linear analysis

Additionally, since the system employs the centralized database connectivity technology, the analysis results are kept in the same database with VBDSPRE, VBDSPST and VBDSTDN. All the analysis results can be visualized through graphs or tables in the foreground while the analysis is processed in the background.

**Preprocessor** is to establish the bridge finite element model. AutoCAD is adopted as the platform for modeling due to its popularity and the state-of-the-art ObjectARX technology. The geometric modeling of the structure can be graphically established through AutoCAD, other than the traditional text-based means. Its main functions are:

- Convert space beam, truss and shell elements from graphic entities
- Generation and editing of the geometric model
- Auto-mesh for wire frame and surface model
- Editing of the bridge construction stages
- Editing of the boundary conditions
- Editing of the multiple load cases
- Editing of the live loads and its moving scope
- Definition of the control target (for cable stayed bridge) and cross sections
• Data exchange between finite element analysis systems

**Postprocessor** is to graphically present the bridge structure analysis results. Besides the preliminary functions of a general-purpose postprocessor, such as internal force and displacement graphs and stress contours, it contains the following special features:

• Organization of the multiple construction stage analysis results
• Graphic representation of the incrementing, accumulative and envelopes results
• Plotting for the influence lines and influence surfaces
• Plotting for the live load envelope
• AASHTO and user definable load combinations
• Cross sectional stress analysis of a shell modeled bridge

**Prestress Tendon Configuration** is to define cross section and prestress tendon profiles graphically and parametrically, and to establish prestress tendon model together with the bridge structural model. It contains the following special features:

• Define the cross sections of a bridge via graphical or parametric methods
• Define the prestress tendon profile via a parametric methods or import from AutoCAD drawings
• Geometry property calculation of a cross section
• Prestress loss calculation
• Define the multiple-stage tensioning schedule
• Build the bridge structural model via the central database
CHAPTER 1 – BRIEF INTRODUCTION

FEA Kernel is specifically designed and developed for bridge structural analysis. Its goal is to provide a generic FEA package for any type of bridges with any possible construction method. It is a general-purpose FEA for bridge engineering and can serve as the kernel of a bridge computer-aided design system.

As the kernel of the bridge computer-aided design system, it provides the engineers a comprehensive bridge structural analysis tool capable of bridge construction simulation. It adopts open system architecture and centralized database to make it easy to be connected with other subsystems such as Preprocessor and Postprocessor. It runs on Windows 2000/98/95/NT platform. Its development is fully exploited the state-of-the-art software technologies like object-oriented programming, graphical user interfacing, visualization, database connectivity and etc.

The analytical functions of FEA Kernel include elastic linear analysis, dynamic modal analysis, elastic stability (class I) analysis, geometry nonlinear analysis with considerations of initial stress and initial strain, and the consideration of the nonlinear sag effectiveness of long cables. The types of finite elements include 3D frame, truss, and shell. The functions and features of the current version are as follows:

- Graphical user interface and visualization of the analysis process
- 3D beam, truss, and shell elements
- Interchange between multiple construction stages
- Analysis of multiple load cases
- Specifically designed, bridge construction-oriented loads
- Automatic adjustment for cable-stayed bridge internal forces
- Construction control analysis
- Prestressing force calculation
- Concrete creep and shrinkage analysis
- Influence surface and extreme live load analysis
- Dynamic modal analysis
- Elastic stability analysis
- Large deformation and other geometry nonlinear analysis

It provides many types of goal to control the automatic adjustment of cables and/or tendons of a cable-stayed structure. For example, some cables can be designated to control the deformation at some locations, while other cables to control the stress at top or bottom flange of a section. For a model that is modeled as shell elements, the goals of tuning cables can even be an integrated internal force over a cross section.

It adopts a new automatic time incremental method to analysis creep and shrinkage of concrete bridge, which is specifically developed with regard to the rapid development of modern computational technologies. The coefficient model includes ACI 209 as recommended in AASHTO LRFD Bridge Design Specifications, CEB-FIP (1990) and CEB-FIP (1978).

The live loading includes all loads specified in AASHTO LRFD Bridge Design Specifications and some other design codes from other countries. It even provides several different types of live load used in railroad bridge design.
The prestress tendons in a bridge are treated as truss elements to simulate its structural behavior, together with the main structure. The displacements of every tendon nodes can be automatically restrained to and established a relationship with its master node in the girder. Several types of loads with regarding to truss element are designed to simulate the tensioning of a tendon. Some stress loss related to the path of tendon, like the loss caused by friction are discrete along its path. While some types of loss, like the loss caused by the creep and shrinkage of the structure during service can be totally obsolete because the tendon itself is already a part of the structure. The computation of the frictional loss is taken into consideration of the current stress status along the tendon, because the part of the friction that is caused by the curvature of the path depends on the stress in the tendon.

Besides the conventional linear elastic analysis, in the geometry nonlinear analysis, the initial stress and initial deformation are taken into consideration during the iteration of large deformation analysis processes. A typical example of initial stress problem is the huge axial force accumulated in the girder of a cable-stayed bridge and a typical example of initial deformation problem is the shape of the main cable of a suspension bridge. The stiffness reduced in the girder of a cable-stayed bridge by the initial stress and the increased in the main cable of a suspension bridge may be critical in live load analysis. Such influences of initial states are taken into consideration not only during the nonlinear iteration analysis due to any dead load case, but also during live load analysis and dynamic modal analysis.

The stability analysis contains two categories. One is linear elastic analysis (Class I), in which only initial stress is considered, and the critical load is solved by evaluation of eigenvalue of the global stiffness matrix and initial stress matrix. The other one is geometry nonlinear analysis with ignoring of the nonlinearity of material. It is not a pure Class II analysis due to the ignorance of material nonlinearity, but it improve a lot compared with the Class I. It’s true that the geometry factor governs the stability in long span bridges. When a bridge is to conduct the second type of stability analysis, initial small turbulence load should be applied to the structure according to different structure. For example, in the lateral stability analysis of the pylon of a cable-stayed bridge, a small lateral force should be applied at the top of the pylon, then to increase the vertical loads along the pylon step by step till the structure fails. The critical load get in this way is quite different from that in eigenvalue analysis and more close to the real condition. In Class I analysis, the stiffness increasing to the pylon in lateral contributed by the initial stress of the cables is taken into consideration only at the pylon’s initial stage, the straight vertical stage. While in the geometry nonlinear stability analysis, the increasing is much higher after the pylon deforms lateral.

In the analysis of creep and shrinkage, it’s not allowed to be combined with initial stress, initial deformation and large deformation analysis.
CHAPTER 2 – THE USER INTERFACE

The following topics are included in this chapter:

- Start VBDS
- Start FEA Kernel
- The menus of FEA Kernel

1 START VBDS AND ITS MAIN SYSTEM INTERFACE

FEA Kernel is integrated with the main system of VBDS. To start FEA Kernel, the main system of VBDS must be started in advance.

To start the main system of VBDS, double click VBDS icon on the desktop. The main window of VBDS will open as shown in figure 1.2.1.

![Main window of VBDS main system](image)

Figure 1.2.1 Main window of VBDS main system

2 SETTING THE PATH OF THE SYSTEM DATABASE

VBDS stores the information that can be shared among different projects in a special database called VBDS system database. The shareable information for bridge analysis and design includes commonly used materials, cross section and etc. The VBDS system database is named as VBDS.MDB when the system is shipped. Before the first starting of VBDS, it is required to set the path of the
VBDS system database.

To set the path name of the VBDS system database, pull down the VBDS menu and select Set system database in the main system window as shown in figure 1.2.1. A dialogue box for selecting the system database opens, as shown in figure 1.2.2. By default setting of the installation package, it locates in the main folder of the VBDS system.

Click Open when VBDS system database is correctly selected.

The command for setting the path of VBDS system database can be issued whenever it is needed, or when ensuring its exact location. Generally, only the first time of setting is required.

3 START FEA KERNEL

3.1 OPEN A PROJECT

Pull down VBDS menu and select Start FEA Kernel in the main system window as shown in figure 1.2.1. A dialogue box for selecting a project database opens, as shown in figure 1.2.3. Usually, the project database is established by Preprocessor. It can also be generated by other subsystems of VBDS.

After the project database file is selected, click Open to start FEA Kernel. FEA Kernel will
start and retrieve data for the structural analysis from the selected project database.

### 3.2 LICENSING VBDS

In the first time of running **FEA Kernel**, **VBDS** needs to be licensed, though a small scale of bridge model still can be analyzed without a license. The licensing of **VBDS** is performed only when the first time to run **FEA Kernel**.

When the first bridge model is retrieved from the project database, a file section dialogue box will open, as shown in Figure 1.2.4. Click **Cancel** to close the dialogue box because this is the first running and now license file had been created.

![Select license file](Figure 1.2.4 Dialogue box for selecting the license File)

A message dialogue box, as shown in figure 1.2.5, will open. Click **Yes** to create a license file related with the computer that is running **VBDS**.

![VBDS](Figure 1.2.5 Dialogue Box for Selecting the License File)

Another file selection dialogue box will open, as shown in Figure 1.2.6. Enter a file name in the **File name** box, and then click **Open**.
A message dialogue box, as shown in Figure 1.2.7 will open. Click **OK** to close the dialogue box. Another message dialogue box, as shown in Figure 1.2.8 will open due to uncompleted licensing.

Email the license file to the author to activate it. When an activated license file is received, follow the procedures in this section again to license **VBDS** in the same computer.

When to license a computer, select the activated license file and click **Open** in the dialogue box as shown in figure 1.2.4. After **VBDS** is licensed, a message dialogue box, as shown in figure 1.2.9, will open. Click **OK** to start **FEA Kernel**.
3.3 START STAGES OF NONLINEAR ANALYSIS

For a bridge model that contains more than one construction stage, if it considers the nonlinear effects of cable sagging or contains some elements that the nonlinear effects of initial stress are considered, the system needs to know from which stage to accumulate the element internal force to calculate these nonlinear effects. This stage is called **Start Accumulating Initial Force Stage**.

For a bridge model that contains more than one construction stage, if it contains any load case that the nonlinearity of initial displacements (a load case that is considered to cause large deformation) is considered, the initial structural configuration, which significantly influences the accuracy of the geometric nonlinear analysis, will be calculated by accumulating displacements caused by any such load cases to the original structure configuration. The stage to start accumulating displacements caused by initial displacements load cases is called **Start Accumulating Initial Displacements Stage**.

For such a bridge model as above, both Start Accumulating Initial Force Stage and Start Accumulating Initial Displacements Stage should be already designated in Preprocessor (preset to the first stage for any model by default, if the situation is other than the default, it should be manually designated). If it is not, FEA Kernel still gives a chance to let user to designate it after the project is opened.

When such a stage needs to be designated by the user, a dialogue box, as shown in figure 1.2.10, will open. Select the stage either in the left list or in the right list in the dialogue box to designate Start Accumulating Initial Force Stage and/or Start Accumulating Initial Displacements Stage, and then click Open to continue.
3.4 THE MAIN SCREEN OF THE BRIDGE FEA KERNEL

When a project is successfully loaded, the main window of the Bridge FEA Kernel will open, as shown in Figure 1.2.11.
3.5 **THE MAIN MENU OF BRIDGE FEA KERNEL**

The top-level pull-down menu of **FEA Kernel** contains: **VBDS, View, Static, Live Load, Construction Control, Dynamic/Stability, Other, Window and Help**.

3.5.1 **VBDS MENU**

**VBDS** menu, as shown in figure 1.2.12, contains the following menu items:

- Set system database  Set VBDS system database
- New Tendon  Start **Prestress Tendon Configuration** with a new project
- Start Tendon  Start **Prestress Tendon Configuration** with an existing project
- Close Tendon  Close **Prestress Tendon Configuration**
- Start FEA Kernel  Start **FEA Kernel**
- Close FEA Kernel  Close **FEA Kernel**
- Start Postprocessor  Start **Postprocessor**
- Close Postprocessor  Close **Postprocessor**
- Exit VBDS  Exit **VBDS** main system

![VBDS menu](image)

**Figure 1.2.12 Menu items in VBDS menu**

**VBDS** menu, as described above, provides the commands to control the entire **VBDS** system. In the main window of **VBDS**, as shown in figure 1.2.1, it has the same **VBDS** menu as in the main window of **FEA Kernel**, as shown in figure 1.2.11.

Only one project can be opened in **FEA Kernel, Postprocessor** and **Prestress Tendon Configuration**. But they can open the same project at the same time. If there is a project is currently opened, the corresponding project-opening menu will become gray as shown in figure 1.2.12.

3.5.2 **VIEW MENU**

**View** menu, as shown in Figure 1.2.13, provides the commands to establish various diagram windows during the analysis process, such as to open or close a window to show the incremental bending moment. It includes:

- Element model  Open/close the window showing only elements and nodes
- Incremental  Open/close a window showing incremental values of a stage
- Accumulated  Open/close a window showing accumulated values up to a stage
Enveloped  
Open/close a window showing enveloped values up to a stage

Stability mode  
Open/close the window showing the stability modal shape

Dynamic mode  
Open/close the window showing the dynamic modal shape

Report general information  
Print a report of the general information of the analysis model

Set the consol font  
Set up the consol font

Incremental, Accumulated and Enveloped menus have same submenu items as below:

**Axial force or longitudinal stress FX**  
Axial forces of beam/truss elements or \( \sigma_x \) of shell elements

**Vertical shear or transverse stress FY**  
Vertical shear forces of beam elements or \( \sigma_y \) of shell elements

**Transverse shear or shear stress FZ**  
Transverse shear forces of beam elements or \( \tau_{xy} \) of shell elements

**Torsion or longitudinal local moment MX**  
Torsional moments of beam elements or \( M_x \) of shell elements

**Transverse moment or transverse local moment MY**  
Transverse moments of beam elements or \( M_y \) of shell elements

**Vertical moment or local torsion MZ**  
Vertical moments of beam elements or \( M_{xy} \) of shell elements

**Displacements**  
Nodal displacements

**Stress of beam or truss**  
Normal stress of beam or truss elements

If the window of a diagram is already opened, the left side of the menu item will show a check mark. If no check mark is on the left, it means that the window is not opened. Select it will open or close the window.

![Menu Items](image)

**Main View menu**

**Incremental/Accumulated/Enveloped submenu**

**Figure 1.2.13 Menu items in View and its sub menus**

### 3.5.3 STATIC MENU

Static menu, as shown in figure 1.2.14, is used to conduct various static analyses. It includes:

**Single stage analysis**  
Conduct single stage analysis one at a time
Multiple stage analysis  Conduct multiple stage analysis at one time
All stage analysis  Conduct all stage analysis at one time
Creep and shrinkage analysis  Conduct creep and shrinkage analysis simultaneously
Creep analysis only  Conduct creep analysis only
Shrinkage analysis only  Conduct shrinkage analysis only

Figure 1.2.14 The Static menu

All stage analysis is to conduct the overall analysis from the first stage to the last stage. For a multiple stage model, if there is only change in certain construction stage or stages after the first analysis, such as readjusting load cases of these stages, Single stage analysis or Multiple stage analysis can be used to save analysis time.

Since the creep analysis is based on the results of all creep shrinkage load case, All stage analysis must be performed before conducting the creep analysis. In VBDS, creep and/or shrinkage are treated as a special load case. The name is always [Creep and shrinkage]. If the analysis conducted is Creep analysis only or Shrinkage analysis only, the results in that load case will include the creep or shrinkage effect only, though the load case name is [Creep and shrinkage]. If the analysis conducted is Creep and shrinkage, results of [Creep and shrinkage] will include the effect of both creep and shrinkage.

3.5.4 LIVE LOAD MENU

Live load menu, as shown in figure 1.2.15, is used to conduct live load analysis. It includes:

Full influence value analysis  Influence value analysis of all elements
Partial influence value analysis  Influence analysis of only selected elements
Loading  Conduct live loading

Figure 1.2.15 The Live load menu

For a small model, a fully influence value analysis will not take significant time and the results will not take too much space. So, it almost makes no difference between the full and the partial analysis. For a large model such as a plate model, however, calculating of the influence values of all elements will take significant time and space. A partial influence analysis will be necessary regarding to save time and space.

These two analysis menus have the same submenus as:

Gravity  Unit vertical gravity influence value for the live loading
Brake  Unit brake influence value for the live loading
Centrifugal  
Unit centrifugal influence value for the live loading

In the current version of VBDS, only the vertical gravity influence value is implemented. The influence value caused by vehicles braking and centrifugal has yet not been released.

3.5.5 CONSTRUCTION CONTROL MENU

Construction control menu, as shown in figure 1.2.16, contains commands to conduct construction control related analysis. It includes:

**Cable force optimization**  
For a cable-stayed model, optimize cable forces according to the definition of goals

**Parameter sensitive analysis**  
Analyze the target values according to a group of element property errors

![Cable force optimization](image1.png)  
![Parameter sensitive analysis](image2.png)

Figure 1.2.16 The **Live load** menu

For the cable-stayed structures, the goals to be optimized can be defined as the moments, the displacements, stresses and equivalent forces of a cross section. **Cable force optimization** will analyze the forces of the tuned cables to reach these goals. For a segmentally erected concrete bridge, guessing the actual material properties, and adopting them in the analysis, is an important procedure of the bridge construction control. Setting a group of errors of material properties, **Parameter sensitive analysis** will analyze the designated goals, so that a further sensitive analysis function can be conducted.

3.5.6 DYNAMIC/STABILITY MENU

**Dynamic/Stability** menu, as shown in figure 1.2.17, contains the commands to conduct dynamic and stability analysis. It includes:

**Elastic stability analysis**  
Perform elastic stability analysis

**Dynamic mode analysis**  
Perform dynamic mode analysis

![Elastic stability analysis](image3.png)  
![Dynamic mode analysis](image4.png)

Figure 1.2.17 The **Dynamic/Stability** menu

For a model that the initial stress effect is concerned, the Class-I stability analysis can be done through the **Elastic stability analysis** command. This kind of stability analysis is archived by solving the eigenvalue problem. Similarly, **Dynamic mode analysis** command will solve the natural dynamic modes. **Elastic stability analysis** and **Dynamic mode analysis** are all based on different stages and different load cases.

3.5.7 OTHER MENU

**Other** menu, as shown in figure 1.2.18, has only one command, **Set sag element initial force**.

![Set sag element initial force](image5.png)

Figure 1.2.18 The **Other** menu
When the sag effect of cables is concerned, the cable forces will influence its stiffness, which will further influence the cable forces. This can be reached through manual iteration. After a round of analysis is complete, this command can be used to save the elements forces of all sag-effect-concerned cables. Thus, the actual stiffness of these cables can be updated in the next round of analysis.

### 3.5.8 WINDOWS AND HELP MENU

Windows menu provides the commonly used Windows command such as Cascade, Arrange, etc. Help menu is for the help system. Figure 1.2.19 shows these two menus.

Figure 1.2.19 The Windows and Help menu

#### 3.6 FEA KERNEL POPUP MENU

Each window of FEA Kernel that shows a diagram of the analysis results has a popup menu to manipulate the graph. Figure 1.2.20 shows the main popup menu and its sub menus.

Figure 1.2.20 The popup menu of FEA Kernel

The way to initiate the menu is to move the cursor to the designated window and then right click the mouse button. The main popup menu includes:

- **View projection**: Set the view projection direction of the graph
- **General attributes**: Set general display attributes of the graph
- **Display filter**: Set element display filter of the graph
- **Text obscure check**: Turn off the overlapped label text
- **Redisplay obscure text**: Turn on all label text
- **Dynamic/Stability mode**: Change dynamic or stability mode attributes, only for dynamic or stability mode window
- **Change stage combination**: Change information associated with stage combination
- **Change stress position**: Change stress point of a section, only for stress window
**Zoom window**  
Zoom the graph by using window method

**Zoom all**  
Display the overall graph

**Zoom out**  
Zoom out the graph

**Zoom in**  
Zoom in the graph

**Dynamic zoom**  
Zoom dynamically

**Pan**  
Pan the graph

**Copy view (WMF format)**  
Copy the graph to Windows clipboard using WMF format

**Copy view (BMP format)**  
Copy the graph to Windows clipboard using BMP format

**Create AVI file**  
Copy the animation graph to an AVI file, only for an animation window

**Windows**  
Windows submenu

**Element select ->**  
Element selection submenu, only for the Element model window

The element selection submenu, as shown in the middle of figure 1.2.20, includes:

**Add**  
Add or remove the selected elements to or from the selection set

**Empty**  
Empty the selection set

**Pick**  
Select element by single pick

**Window**  
Select element by using window method

**Cross window**  
Select element by using cross window method

**Fence**  
Select element by using fence method

**Other**  
Select element by using other non-graphical method

The windows submenu, as shown in the right of figure 1.2.20, includes the common window operations, which are the same as other Windows application.

All the graphs showing in the FEA Kernel are projected from the real 3D model space. By default, the project direction, which can be adjusted by using View projection, is parallel to the Z-axis.

**General attributes** of a graph include different display colors, widths, nodal symbols, sizes, text fonts, text sizes, scale factors for the results and etc.

For a stability or dynamic modal shape graph window, its attributes including stage and load case can be changed by Dynamic/Stability mode. A mode shape graph can be displayed as animation and the animation can be copied to an AVI file, which can be played independently in Windows media player.

**Display filter** can filter some elements off screen, so that only the interested elements are showing in the window.

If labels of a graph are overlapped, use Text obscure check to turn off the overlapped text. In some cases, this command may need to be executed twice to completely turn off the overlapped text. The turned off labels can be turned on by using Redisplay obscure text.

Any kind of graph showing some type of analysis results, is of a certain stage and of a certain
load case, or combined from a group of stages and load cases. For example, an incremental graph relates to a stage and a load case; an accumulated or an enveloped results relates to a stage combination schedule, which specifies which load case of which stage will participate at how much a load factor. Use **Stage combination** to change the combination schedule of a graph.

For beam/truss elements, not only the forces, but also the normal stresses can be displayed. VBDS predefines 8 points over the cross section of beam/truss elements. By setting the sectional and material modulus of each point, stresses of totally 8 points can be displayed. The 8-point can be used to simulate different layers of a structure, such as the concrete deck at the top, the rebar, the bottom flange and etc. Use **Change stress position** to change the stress point of a section.
CHAPTER 3 – STATIC AND LIVE LOAD ANALYSIS

The following topics are included in this chapter:

- Some concepts of FEA Kernel
- Static analysis
- Concrete creep and shrinkage
- Live load analysis

1 SOME CONCEPTS OF FEA KERNEL

As a general FEA system for bridge structures, FEA Kernel provides the facilities for simulating the bridge construction. Its main functions include multiple stages and multiple load case analysis, concrete creep and shrinkage analysis and live load analysis.

The static analyses are based on stages. It can be conducted as one-stage analysis, multiple-stage analysis and all-stage analysis. When a stage is to be analyzed, all load cases of that stage will be analyzed. The concrete creep and shrinkage analysis can be conducted as creep-only analysis, shrinkage-only analysis and both-creep-shrinkage analysis. The live load analyses include the influence line/surface evaluation and live loading.

The static analysis can be linear or nonlinear. By default, the analysis of any load case at any construction stage is linear. It will be treated as nonlinear and the analysis will be iterated if the load case is an initial-displacements-considered load case or the model does exist some initial-forces-considered elements without specifying their initial forces. Otherwise, the static analysis of a load case will be conducted as linear.

Even if the analysis of a load case, the analysis of creep and shrinkage and the influence value are structurally linear, some of the nonlinear effects are still considered during the analysis:

- The geometric configuration has already been updated to the current stage according load case history. This feature allows conducting a static analysis of a particular load case after a suspension bridge is complete. For example, a cable can be built starting from zero stress, and then some load case will act on it to increase its stresses. At the final stage, some minor load case can be analyzed as linear at the current cable shape.
- The stiffness has already been considered the sag effects of cables and the P-Δ effects due to initial stresses statically. This feature allows considering the girder initial stress and the cable sag during the influence value evaluation after a cable-stayed is complete.

The following are some tips for understanding FEA Kernel:

- A model can have multiple construction stages. One stage can have unlimited load cases. When a stage is analyzed, its all load cases are analyzed.
- The analysis of a stage can be repeated any times.
- If some load cases of some stages are changed after a full analysis, these modified load cases can be re-analyzed, so that to update the results, by conducting the analysis of these stages only.
- Live loading can only be conducted after the influence values are evaluated. However,
it is not necessary for doing both at one analysis session.

- The influence values are evaluated at the service stage of a bridge, which is designated among all stages.

- One analysis of a load case includes assembling the global stiffness matrix, assembling the global initial stresses matrix, factorizing the matrix, building the force vector of all load cases, evaluating displacements and forces, saving results to database and updating the real-time graphs.

- Creep and shrinkage can only be conducted after all stages’ static analyses are done. It is also not necessary for doing both at one analysis session.

- Creep and shrinkage analysis depends on the construction sequences, concrete ages, elements’ creep and shrinkage properties and the loading histories of all creep-shrinkage-considered load cases.
CHAPTER 3 – BRIDGE FEA PREPROCESSOR

3.1 Preprocessor Installation

The Bridge FEA Preprocessor subsystem is formed under AutoCAD. It is necessary to initiate the AutoCAD before initiating the Preprocessor.

Input `appload` under Command: to install the Preprocessor.

Command: `appload`

After inputting the command, screen show the dialogue box as illustrated in Figure 3.1

Figure 3.1 Dialogue Box after installing the Preprocessor

When the Preprocessor is initiated at the first time, it is necessary to press the File key inside the dialogue box to select the Preprocessor execution mode. After pressing the File key, the dialogue box is illustrated in Figure 3.2. The execution mode documentation of the Preprocessor, named preprocessor.arx, is kept in the BRCAD system catalog.

Once established, when initiating the Preprocessor again, user can select preprocessor.arx document directly from the dialogue box document list.

Figure 3.2 Dialogue Box of selecting the Preprocessor Execution Document

After selecting the preprocessor.arx inside the dialogue box document list as illustrated in Figure 3.1, the corresponding Load key becomes visible. Hit the key to install the Preprocessor subsystem.

3.2 Preprocessor Initiation

After loading Preprocessor subsystem to the AutoCAD system, it is necessary to initiate a new or open an old finite element model database.

3.2.1 Establishing a new finite element model database
The existing subsystems of BRCAD are formed by the unified core database. This means that the subsystems of the Preprocessor, the FEA and the Postprocessor are using the same project database. This project database is installed by establishing the new finite element model database inside the Preprocessor subsystem.

To establish a new finite element model database, input the following command following the AutoCAD command symbol:

Command: newpre

_Newpre_ is a new graphic system command after installing the Preprocessor subsystem to the AutoCAD. After typing in the command, a dialogue box as illustrated in Figure 3.3 is shown:

![Figure 3.3 Dialogue Box of Inputting a New Finite Element Model Name](image)

After selecting a proper client catalogue, input the finite element model database document name under the name command. Press the **Open** button to represent establishing a new database. Then the screen shows the dialogue box as illustrated in Figure 3.4.

![Figure 3.4 Dialogue Box of Selecting BRCAD System Databases](image)

Besides the project database, user has to designate the BRCAD system database. The system database keeps the commonly used material properties, which are required by the finite element analysis. This system database is provided by the BRCAD system. It is installed to the BRCAD system catalogue while installing the system. After selecting the system database, press the **Open** button to initiate the Preprocessor.

### 3.2.2 Opening an existing finite element model database

The command to open an existing finite element model database is **openpre**. The process of inputting and executing is similar to establish a new finite element model database. The difference is that the database must be in existence while selecting a project database.

### 3.3 Preprocessor Interface

#### 3.3.1 Main interface of the Preprocessor
While inputting `newpre` to establish a new finite element model database or `openpre` to open an existing one under the AutoCAD system command, the Preprocessor subsystem is initiated. Then, the AutoCAD interface is switched to the Preprocessor main interface as shown in Figure 3.5.

Figure 3.5 The Main Interface of the BRCAD Preprocessor Subsystem

The Preprocessor and the AutoCAD are integrated as one. The windows and the pull-down menus of the Preprocessor and the AutoCAD are sharing the same main system frame. While the mouse activates the AutoCAD main window, the system pull-down menu is automatically switched to the AutoCAD pull-down menu. Then, user can input the AutoCAD keyboard command inside the command zone. While the mouse activates the Preprocessor, the system pull-down menu is automatically switched to the Preprocessor pull-down menu. Then the AutoCAD keyboard command is deactivated. Inside the Preprocessor, there are many commands which need picking up the geometric parameters from the AutoCAD window. If the active window is not AutoCAD, user has to activate the AutoCAD main window before making the selection.

3.3.2 Formation of the Preprocessor main window

Based on their functions, the main window of the Preprocessor is divided into the database listing window, the database table window, the system word output window as shown in Figure 3.6.

The database list window is used to present names of the table, the relation, and the SQL inquiry. As shown in Figure 3.7, the table is processed through a tree structure according to the BRCAD core database, the project database, and the order of the table/relation/SQL inquiry.

Figure 3.6 Formation of the Preprocessor Main Window

Figure 3.7 The Formation of the Database Table Listing Window

While listing tables, user merely moves the pointer to the item with the + sign and then double click the left key of the mouse. Next layer of the sub-item belonging to the pointed item will be popped up. If the pointer is moved to the item with a – sign and double clicked, the next layer of that item will be closed. If the item has no + or – sign, it means that item has no sub-item.

While moving the pointer to any item, press the right key and a menu will be popped up. Through this menu, user can execute different operation of many databases, such as opening a database table, establishing or executing an inquiry or SQL, etc. Please refer to the section containing the Formation of the Preprocessor Pop-up Menu.
Through the related commands inside the pop-up menu of the execution database table window, user is able to open the database table or results of the inquiry. They are all listed inside the database table presentation. To distinguish the difference, the database table and the inquiry result can be shown in different colors. In addition, the database can be modified whereas the inquiry result cannot. To modify the database table, it is required to move the pointer to the record that needs to be modified and activate it. Then, press the right key to pop up a menu. Edit the table record through the related command of this menu. Please refer to the section containing the Formation of the Preprocessor Pop-up Menu.
3.4 **Formation of the Preprocessor Pull-down Menu**

There are eleven items in the Preprocessor pull-down menu and they are: System, Create, Edit, Regenerate, Stages, Control, Others, Display, Exchange, Window and Help.

### 2 **System Menu**

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set system database</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Preprocessor workspace</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Close project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reload AutoCAD Menu</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

System menu is used to set the system of the Preprocessor, which includes:

- **Set system database**: Set system database of the current project
- **Preprocessor workspace**: Set workspace for the Preprocessor
- **Close project**: Close the current project
- **Reload AutoCAD Menu**: Reload AutoCAD menu instead of BRCAD menu

### 3 **Create Menu**

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beam by 2 points</td>
<td>Beam by entity or points</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Truss by 2 points</td>
<td>Truss by entity or points</td>
<td>Simple bearings</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 Nodes triangle flat shell</td>
<td>6 Nodes triangle flat shell</td>
<td>4 Nodes rectangle flat shell</td>
<td>8 Nodes quadrilateral flat shell</td>
<td>Flat shell from edges or entity</td>
<td>Current parameters</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
“Create” menu covers all the commands associated with element generation, including:

- **Beam by 2 points**: Create space beam element through two points.
- **Beam by entity or points**: Create space beam element through entity or geometric points.
- **Truss by 2 points**: Create space truss element through two points.
- **Truss by entity or points**: Create space truss element through entity or geometric points.
- **Simple bearings**: Create truss element as bearing at the ends of existing space beam or truss element.
- **3 nodes triangle flat shell**: Create three-node triangular shallow shell element.
- **6 nodes triangle flat shell**: Create six-node triangular shallow shell element.
- **4 nodes rectangle flat shell**: Create four-node rectangular shallow shell element.
- **8 nodes triangle flat shell element**: Create eight-node quadrilateral shallow shell element.
- **Flat shell from edge or entity**: Create shallow shell from edge(s) or entity of curved surface(s).
- **Current parameters**: Set current parameters generated by the element.

Each element has its own material symbol number, cross section symbol number and reference vector symbol number. The Preprocessor has set default numbers for the element. When the element is generated, these values will be automatically adopted. These default values may be reset by the “Current parameters” command. These default values will automatically be recorded to the database.

**4 Edit Menu**
“Edit” menu covers all the commands for the element editing, including:

- **Move element**: Move the element(s) in space
- **Rotate element**: Rotate the element(s) in space
- **Mirror element**: Create a mirror image of the element(s) in space
- **Assemble element**: Assemble the element(s) in space
- **Move + Copy element**: Move and copy the element(s)
- **Rotate + Copy element**: Rotate and copy the element(s)
- **Mirror + Copy element**: Create a mirror image and copy the element(s)
- **Assemble + Copy element**: Assemble and copy the element(s)
- **Delete element**: Delete the element(s)
- **Divide element**: Divide an element to elements
- **Move node**: Move node(s) of an element
- **Element grouping**: Separate element(s) to groups
- **General attributes**: Edit attributes of an element(s)
- **Assign creep property**: Assign element creep properties
- **Unify local CS**: Unify local creep and shrinkage of the element(s)

The “Rotate element” and “Mirror element” commands are different from those of a geometric entity. The rotation axis of an element is a straight line in space, not like the rotation command in
AutoCAD, which passes a point normal to a straight line of a current working XY plane. The mirror plane of an element is also any plane in space, which is also different from AutoCAD.

Also, the meaning of “Copy element” command is to create a new element at a new position but not changing the original element. All the entities of the new element, such as element type, material properties, cross section properties and reference vectors, are the same, but with different number and at different position.

“Assemble element” command is used to assemble a group of elements in space. Assembling element is similar to assembling basic pieces of a model. Its principal theory is to define a local coordinate system of a group of elements and then define this local coordinate system at the new position in space. By taking advantage of this function, user can easily assemble the subsystem of an element model.

“Move node” command is to move the defined nodal point of an element. Due to the fact that an element and its nodes are corresponding each other, moving the defined nodal point of an element is to modify the element position.

After the element is created and there is a need to modify the entity of the element, it can be done through the following commands in the submenu of Generate attribute:

- Modify material ID
- Modify primary section ID
- Modify reference vector ID

Modify the material identification number of an element
Modify the primary section identification number of an element
Modify the reference vector identification number of an element

5 Regenerate Menu

“Regenerate” menu is to regenerate the corresponding finite element model based on the AutoCAD entity from the model data in the finite element database or the AutoCAD drawing file (DWG).

Element geometric model is contained in the finite element model database and the AutoCAD
drawing file (DWG) used in the preprocessor. However, the information contained in the finite element model database is all for the finite element model. The database contains the complete finite element information, as well as the drawing information, such as the drawing layers and colors. On the contrary, the drawing file, besides element ID, type and nodal locations, contains incomplete information associated with finite element analysis, such as element material properties, primary section, etc. Once the model is generated, DWG file can be deleted.

When either the FEA database or the DWG file is damaged or lost, it is possible to recover partial information of the FEA database or DWG file from the other side. Based on the FEA database the element geometric model can be recovered but not the attributes like the model’s layers and colors. On the other hand, based on the DWG file to recover the FEA database, element list and nodal table can be recovered but not the material properties, cross section attributes and vector references.

Regenerate menu includes two menu items:

- **Regenerate entity**
  - Regenerate element graphic entity based on the database
- **Regenerate data table**
  - Regenerate database element nodal table based on the DWG file

Sometimes during the regeneration, due to improper operation or short circuit, the linkage between FEA database and DWG file appears incompatible, such as the case when through graphic entity picking up element, the system cannot recognize the element member though the graphic entity. In this case, it is recommended to delete the element entity in the DWG model and then regenerate the graphic entity.

### 6 Stages Menu

The Preprocessor is a bridge finite element preprocessor developed for multiple stages. Each
construction stage is an independent finite element model but different construction stage can be composed by the same elements. Stage editing means generation of a construction stage, element composition within a designated construction stage, and restrained condition of a construction stage. Menu for the editing of the construction stages includes:

- **Element construction**: Edit element construction of a construction stage
- **Element’s section ID in stage**: Edit element section ID in a construction stage
- **Connectivity threshold and sorting order**: Set automatic connectivity threshold and sorting order of missing element nodes
- **Dnode boundary**: Edit boundary conditions of a construction stage
- **Rigid connection**: Edit rigid connection of a construction stage
- **Create Fnode Number**: Create finite element nodal numbering of a construction stage
- **Reinitialize Dnode boundary**: Reinitialize boundary condition of a construction stage
- **Auto-generate multi-stage boundary**: Automatically generate boundary conditions of multiple stages
  - **Stage’s general information**: Edit general information of a construction stage
  - **Set usage stage**: Set usage stage

The submenu for the Element constitution command includes:

- **Add element to stage**: Add element to a construction stage
- **Remove element from stage**: Remove element from a construction stage

The submenu for the Dnode boundary command includes:

- **Fix/Free restraint**: Edit Fix or Free boundary condition
- **Inherit Fix/Free restraint**: Inherit Fix or Free boundary condition from other stage
Master-slave restraint Edit master-slave restraint condition
Add joint Add joint between elements

The submenu for the Rigid connection command includes:

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Element constitution</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Element’s section ID in stage</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Welding threshold and sorting order</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>DNode boundary</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Rigid connection</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Create FNode Number</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Reinitialize DNode boundary</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Auto-generate multi-stage boundary</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Stage’s general information</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Set usage stage</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Automatic rigid connection</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Manual rigid connection</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Remove rigid connection</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Automatic rigid connection Automatically generate rigid connection of a construction stage
Manual rigid connection Manually generate rigid connection of a construction stage
Remove rigid connection Remove rigid connection of a construction stage

7 Control Menu

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Element error parameter</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Control target description</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Control target value</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Edit tuning cable</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Backward/forward stage reference</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Construction “Control” menu is used to set the element error parameters, control target and cable adjustment, which includes:

Element error parameter Edit construction error parameters of the element properties
Control target description Edit control target description
Control target value
Edit control target value
Edit turning cable
Edit cable element needing to be adjusted for the cable stayed bridge
Backward/forward stage reference
Set the corresponding reference of the backward and forward construction stages for analysis

8 Other Menu

Other menu is used to edit load cases, element reference vector, unit force location, etc. The menu includes:

- Load case
  - Edit load cases
- Loading schedule
  - Edit schedule of each loading
- Reference vector
  - Edit element reference vector
- Creep property
  - Edit element creep properties
- Unit force location
  - Edit the unit force location
- Live load region
  - Edit active region for the live load
- Live load description
  - Edit descriptive information associated with live load
- Default value of triangle net
  - Set default value of the triangle net of the influence surface
- Sagged cables
  - Set cable element requiring consideration of the self-weight sagging for the cable stayed bridge
- Initial stress concerned element
  - Set element requiring consideration of the initial force or stress
Cross section definition - Define element cross section for the stress analysis
Cross section related element - Define element composition of the cross section

9 Display Menu

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Elements of a stage</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Element label</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Element LCS</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>DNode boundary</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Display rigid connection</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Turn off rigid connection</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Display menu is used to display the element number, the boundary conditions, etc. The menu includes:

- Elements of a stage - Display elements of a construction stage
- Element label - Display the element label
- Element LCS - Display element Local Coordinate System (LCS)
- Dnode boundary - Display boundary constraints of a construction stage
- Display rigid connection - Display rigid connection
- Turn off rigid connection - Turn off the display of the rigid connections

10 Exchange Menu

<table>
<thead>
<tr>
<th>System</th>
<th>Create</th>
<th>Edit</th>
<th>Regenerate</th>
<th>Stages</th>
<th>Control</th>
<th>Other</th>
<th>Display</th>
<th>Exchange</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Import from standard data file</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Export to standard data file</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

“Exchange” menu is used to exchange information with other FEA software, such as transforming the finite element model to a SAP input file, or input finite element model from SAP input file. Currently, the exchange ability is to import and export element nodal model from and to the database. The corresponding submenus are:

- Import from standard data file - Import element nodal model from the standard data file
- Export to standard data file - Export to standard data file from the element nodal model of a
construction stage

11 **Window Menu**

[Menu options and icons]

12 **Help Menu**

[Menu options and icons]

Window menu is used for the usual Windows operation and Help menu is used to initiate the help system for the BRCAD. They are similar to the other Window and Help menus and are not further discussed.
3.5 Formation of the Preprocessor Pop-up Menu

There are two pop-up menus for the Preprocessor and they are the database table listing window pop-up menu and the database table presentation pop-up menu. The former is used to execute commands such as opening database table. Later is used to execute commands like modifying the database table.

The way to pop up the database table listing window is to move the pointer to a certain item inside the database table listing window and press the right key of the mouse. The way to pop up the database table presentation is to use the mouse to activate any cell or group of cells (including the record title block), and then press the right key of the mouse.

13 Database Table Listing Window Pop-up Menu

<table>
<thead>
<tr>
<th>General information</th>
<th>Display general information of the Preprocessor, project database or system database in the text output window</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compact database</td>
<td>Compact project database</td>
</tr>
<tr>
<td>Repopulate table list</td>
<td>Repopulate the database table list</td>
</tr>
<tr>
<td>Show table schema</td>
<td>Show the table schema in the text output window</td>
</tr>
<tr>
<td>Show table data</td>
<td>Show the table data in the text output window</td>
</tr>
</tbody>
</table>

The database table listing window pop-up menu includes the following submenus:
<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Show field schema</td>
<td>Show the field schema of the table in the text output window</td>
</tr>
<tr>
<td>Show relation</td>
<td>Show the relation schema in the text output window</td>
</tr>
<tr>
<td>Show index</td>
<td>Show the index schema in the text output window</td>
</tr>
<tr>
<td>Show SQL</td>
<td>Show the query or the SQL schema in the text output window</td>
</tr>
<tr>
<td>Edit SQL</td>
<td>Edit the query or the SQL sentence</td>
</tr>
<tr>
<td>New SQL</td>
<td>Generate new query or SQL sentence</td>
</tr>
<tr>
<td>Delete SQL</td>
<td>Delete the query or the SQL sentence</td>
</tr>
<tr>
<td>Execute query</td>
<td>Execute query and display the query result in the text output window</td>
</tr>
<tr>
<td>Execute SQL</td>
<td>Execute SQL sentence</td>
</tr>
</tbody>
</table>

After popping out the menu, above submenu items will become gray based on the active listing items. When some of the pop-up submenu items become gray, it means that the gray commands of the current listing items can not be executed, such as the case that if the current active list item is the Table names, all the submenu items on the pop-up menu except General information, Compact database and Repopulate table list, all become gray.

**Database Table Presentation Pop-up Menu**
CHAPTER 4 – BRIDGE FEA POSTPROCESSOR

4.1 BRCAD Initiation and Main System Interface

Due to the fact that there is no need to be supported by the CAD system, the Postprocessor subsystem and the FEA Kernel subsystem are formed under the BRCAD system. These two subsystems are using the same interface. It is necessary to initiate the BRCAD before initiating the Postprocessor. The way to initiate the BRCAD is described in Section 2.1 - BRCAD initiation and main system interface.

4.2 Postprocessor Initiation

In the BRCAD main system pull-down menu, single click “Start Postprocessor” to initiate the Postprocessor subsystem. The dialog box instruction is the same as the one for the FEA Kernel to select the project database and system database. Additionally, if there is no definition of the construction stage in the Preprocessor, the system, similar to the initiation of the FEA Kernel, will ask the user to select the construction stage.

After selecting the structural analysis project database and the system database, the Postprocessor subsystem starts to read the basic data from the database, which is similar to the FEA Kernel. After finishing reading the data, the Postprocessor subsystem main window will automatically generate two child windows as shown in Figure. 4.1. One of the child windows is the element model windows and the other is the text report output window.

Element model window is used to display element models of various construction stages. In the mean time, this window is used to select the element subset required to be processed and to set the load combination of the construction stage. The text report output window is used to display all the stress and displacement results expressed in text. These two child windows are two important and fundamental windows of the Postprocessor. If they are closed, it is necessary to open them by the command shown in the menu before other Postprocessor operation for a normal process.
4.3 Formation of the Postprocessor Pull-down Menu

The Postprocessor pull-down menu is formed by eleven items: BRCAD Main, Setup, Displacements, Forces, Stress, Influence, Live Load, Usage, Combination, Windows and Help.

In the above menu items, BRCAD Main, Windows and Help are mutually used with the FEA Kernel. The description is referred to Section 2.3 - Formation of the FEA Kernel Pull-down Menu.

13.1 SETUP MENU

<table>
<thead>
<tr>
<th>BRCAD</th>
<th>Setup</th>
<th>Displacements</th>
<th>Forc es</th>
<th>Stress</th>
<th>Inf.</th>
<th>Live load</th>
<th>Usage</th>
<th>Combination</th>
<th>Windows</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Open text report</td>
<td>Open element model</td>
<td>Set text report font</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Setup menu includes the following three items:

Open text report Open the text report output window
Open element model Open the element model display window
Set text report font Set up font for the output report
Due to the fact that the text report output window and the element model display window are two fundamental display windows of the Postprocessor subsystem, openings of many graphic functions need these two windows. If these two windows are closed, they can be reopened by the command in the Setup menu. If these two windows are already opened, the corresponding menu items will automatic show gray.

### 13.2 DISPLACEMENT MENU

Due to few requests of rotations, Displacement menu does not include rotations.

Before displaying displacements, the element set in the element model window has to be selected. Only displacements of those selected elements can be shown. Additionally, static loading and loading...
combinations of construction stages are required to be set through the popup menu in the element model window.

### 13.3 FORCE MENU

Mirrored to the Displacement menu, the Force menu is used to display the internal forces generated by the static force, including incremental forces generated by certain static loading in a certain stage, accumulated forces and enveloped forces generated by certain static loading to a certain stage. Force menu includes following menu items:

- **Incremental**
  - Incremental force submenu
- **Accumulated**
  - Accumulated force submenu
- **Enveloped**
  - Enveloped force submenu
- **Settlement envelope**
  - Force envelope due to support settlement subsystem
- **Temperature**
  - Temperature force envelope subsystem

The submenus of the above menu items are all the same, including:

- **Axial force or longitudinal stress FX**
  - Axial force of the beam element or $\sigma_x$ of the shell element
- **Vertical shear or transverse stress FY**
  - Vertical shear of the beam element or $\sigma_y$ of the shell element
- **Transverse shear or shear stress FZ**
  - Transverse shear of the beam element or $\tau_{xy}$ of the shell element
- **Torsion or longitudinal local moment MX**
  - Torsion of the beam element or $M_x$ of the shell element
- **Transverse moment or transverse local moment MY**
  - Transverse moment of the beam element or $M_y$ of the shell element

- **Temperature envelope**
  - Temperature force envelope subsystem

- **Major principal stress**
  - Major principal stress
- **Minor principal stress**
  - Minor principal stress
Vertical moment or local torsion $M_Z$, vertical moment of the beam element or $M_{xy}$ of the shell element

Major principal stress $\sigma_1$ of the shell element

Minor principal stress $\sigma_2$ of the shell element

Before displaying forces or stresses, the element set in the element model window has to be selected. Only forces or stresses of those selected elements can be shown. Due to the fact that different types of element are showing forces or stresses in different ways, usually the types of the selected elements should be the same. Especially, the output of shell element stresses and beam element forces cannot be mixed together. Additionally, static loading and loading combinations of construction stages are required to be set through the popup menu in the element model window.

The major and minor principal stresses shown in the menu are only for the shell elements.

When the structural analysis adopts the shell elements, the force results are local stresses and local moments on the element nodes, not the cross section forces for the engineering design use. In addition, the stress contours shown on the shell element plane are not able to show the influence of the cross section shear lag and twisting angle. The Postprocessor provides the function to analyze the cross section of the shell analysis model. Its main function is to display output cross section stress distribution and calculate equivalent forces through numerical integration. Based on the equivalent force, the average stress of the cross section can be calculated by following the basic beam theory. With the current version, only the static stress can be calculated. Live load stress analysis is not covered in the current version.

14 Influence Menu

<table>
<thead>
<tr>
<th>BR/CAI</th>
<th>Setup</th>
<th>Displacements</th>
<th>Forces</th>
<th>Stress</th>
<th>Inf.</th>
<th>Live load</th>
<th>Usage</th>
<th>Combination</th>
<th>Windows</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Displacements</td>
<td>Forces</td>
<td>Stress of beam or truss</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Influence line/surface menu is used to display the output force or displacement influence lines or surfaces, including the following submenus:

- Displacements
  - Displacement influence lines or surfaces
- Forces
  - Force influence lines or surfaces
- Stress of beam or truss
  - Stress influence lines or surfaces of beam or truss
Live load Menu is used to display the live load force or displacement envelope, including the following submenus:

- **Displacements**
  - Live load displacement envelope
- **Force**
  - Live load force envelope
- **Stress of beam or truss**
  - Live load stress envelope of beam or truss

Submenus of displacement menu include following items:

- **Longitudinal DX**
  - Live load displacement envelope in X direction
- **Vertical DY**
  - Live load displacement envelope in Y direction
- **Transverse DZ**
  - Live load displacement envelope in Z direction

Submenus of the live load force menu include following items:

- **Axial force or longitudinal stress FX**
  - Envelope for the axial force of the beam element or \( \sigma_x \) of the shell element
- **Vertical shear or transverse stress FY**
  - Envelope for the vertical shear of the beam element or \( \sigma_y \) of the shell element
- **Transverse shear or shear stress FZ**
  - Envelope for the transverse shear of the beam element or \( \tau_{xy} \) of the shell element
Torsion or longitudinal local moment MX
Envelope for the torsion of the beam element or $M_x$ of the shell element

Transverse moment or transverse local moment MY
Envelope for the transverse moment of the beam element or $M_y$ of the shell element

Vertical moment or local torsion MZ
Envelope for the vertical moment of the beam element or $M_{zy}$ of the shell element

16 Usage Menu

Usage menu is used to generate or delete forces or displacements of certain usage stage, i.e. based on the analysis results from the FEA Kernel to generate or delete non-live load report for forces of usage stage or displacements of usage stage. This menu includes two submenus:

Generate
Generate non-live load item for forces of usage stage and displacements of usage stage

Delete
Delete non-live load item for forces of usage stage and displacements of usage stage

The submenus have the same contents:

Gravity displacements
Displacements due to structural self weight

Prestress displacements
Displacements due to prestress loading

Creep and shrinkage displacements
Displacements due to concrete creep and shrinkage

Temperature displacements
Displacements due to temperature changes
Settlement displacements  Displacements due to support settlements
Gravity forces  Forces due to structural self weight
Prestress forces  Forces due to prestress loading
Creep and shrinkage forces  Forces due to concrete creep and shrinkage
Temperature forces  Forces due to temperature changes
Settlement forces  Forces due to support settlements

17 **Combination Menu**

The load combination menu is used to edit load combination coefficient table, generate, delete or display combination results, including following submenu items:

<table>
<thead>
<tr>
<th>Combination coefficient table</th>
<th>Edit combination coefficient table</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generate</td>
<td>Generate force and displacement combinations based on the designated load combinations</td>
</tr>
<tr>
<td>Delete</td>
<td>Delete the designated load combination results</td>
</tr>
<tr>
<td>Display</td>
<td>Display the designated load combination results</td>
</tr>
</tbody>
</table>

Generate and Delete menus have the same submenus:

<table>
<thead>
<tr>
<th>Displacements</th>
<th>Displacement combinations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forces</td>
<td>Force combinations</td>
</tr>
</tbody>
</table>
Display menu includes the following submenu items:

- **Longitudinal DX** Displacement combination results in X direction
- **Vertical DY** Displacement combination results in Y direction
- **Transverse DZ** Displacement combination results in Z direction

Axial force or longitudinal stress FX Combination results for the axial force of the beam element or $\sigma_x$ of the shell element

Vertical shear or transverse stress FY Combination results for the vertical shear of the beam element or $\sigma_y$ of the shell element

Transverse shear or shear stress FZ Combination results for the transverse shear of the beam element or $\tau_{xy}$ of the shell element

Torsion or longitudinal local moment MX Combination results for the torsion of the beam element or $M_x$ of the shell element

Transverse moment or transverse local moment MY Combination results for the transverse moment of the beam element or $M_y$ of the shell element

Vertical moment or local torsion MZ Combination results for the vertical moment of the beam element or $M_{xy}$ of the shell element

### 4.4 Postprocessor Pop-up Menu

The Postprocessor pop-up menu is popped up by hitting the right key of the mouse in the element model window and various force or displacement output windows. The contents of the pop-up menu are different following the display windows and can be divided into four types. The first type is the menu popup in the element model windows. The second type is the menu popup in the non-contour windows. The third type is the contour window. The forth type is the stress window of the shell elements.
Compared to the FEA popup menu shown in Section 2.4, the first type popup menu adds one more menu item:

Generate text reports  
Output to the text report window with what shown in the graphic window

Other menu items are the same as the ones shown in Section 2.4 – FEA Popup Menu. The other three types either add or delete some items from the first type.

In the non-contour windows, such as forces, displacements, influence lines of the beam elements, the pop-up menus, compared to the first type, deletes the following items:

Display filter  
Set element display filter of the graph
Change stage combination  
Change information associated with stage combination
Element select  
Element select submenu

In the contour menu, such as influence surface, the popup menu adds the following item:
Contour attributes | Set contour display attributes

In the popup menu of the stress window for the shell element, one more item is added:

- **Change stress LCS** | Change the unified stress Local Coordinate System (LCS) of the shell elements

In addition, the text report window has its own popup window. The way to pop up the menu is the same as the other popup menus. Its contents are:

- Add
- Empty
- Pick
- Window
- Cross window
- Fence
- Aperture
- Other method