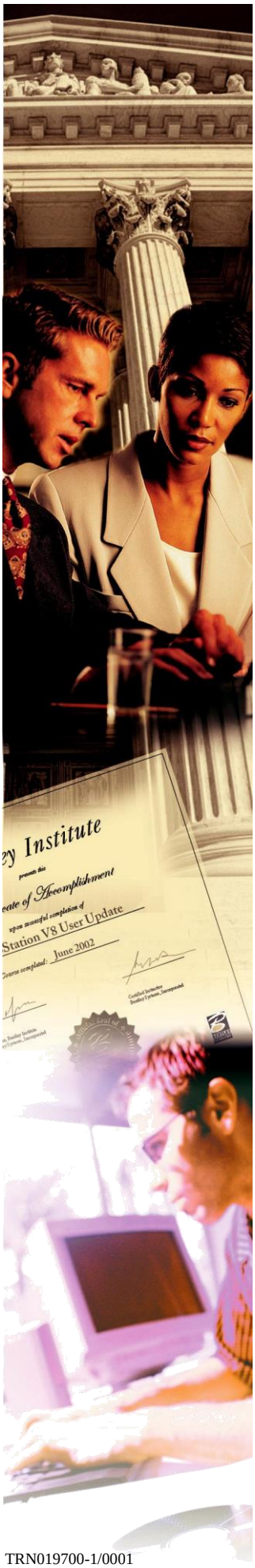


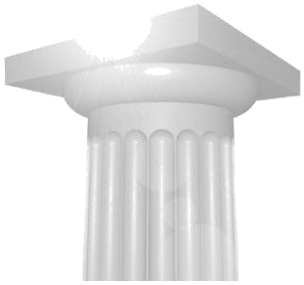
SACS 拖航分析

SACS Connect Edition

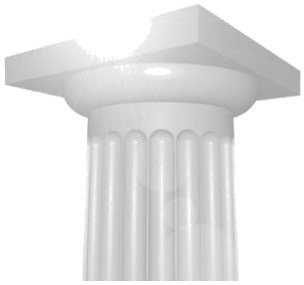
孟文　　Bentley软件
Kevin.meng@bentley.com

TRN019700-1/0001

Bentley Institute



目录



课程总览

课程介绍

本章将学习如何在SACS进行拖航分析

目标用户

本课程是用户一下目标客户:

- 结构工程师

课程目标

完成本课程后，你将具备以下能力:

- 在SACS中完成一个拖航分析

课程准备

- 熟悉各种结构设计理论以及规范

软件准备

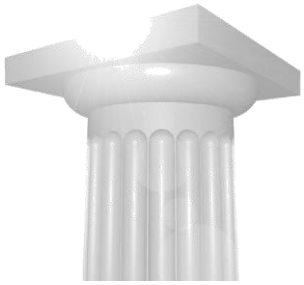
此课程内容依据一下软件版本编写:

- SACS 11.0 Connect Edition

课程内容

此课程包含以下内容:

- 拖航分析



拖航分析

拖航分析

准备

1 创建一个拖航分析的工作目录

2 将装船Copy the jacket model file, **sacinp.jkt** and the seastate input file, **seainp.loadout** from load-out directory into the transportation directory.

3 Make the directory as the current working directory.

General Information

A complete tow analysis includes the following parts:

- **Gap analysis for dead load**: In this static analysis only dead loads are applied on the jacket structure. The jacket structure is supported by all eight vertical supports.
- **Tow analysis**: The tow analysis for transportation inertia loads only. The structure is supported by all eight vertical supports and all tie-down members.
- **Combine analysis**: Combine the results from the above two analyses to generate a combined common solution file.
- **Post processing**: Perform member code checks and joint can checks, and create the postvue database.

➔ **Exercise: Modifying model file sacinp.jkt to add tie-down members**

It is assumed that all vertical support CAN members will be effective during jacket load-out and tow, and all tie-down members will be effective only during tow.

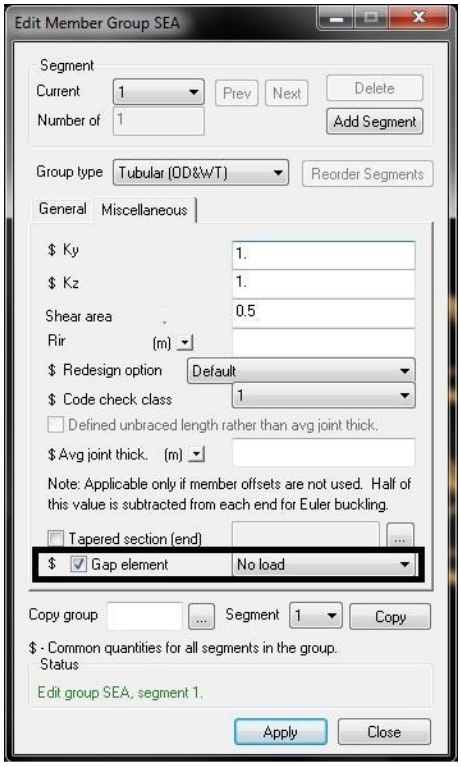
1 Add joints on barge deck level at Elevation **0.0**, and name them **T111**, **T112**, **T121**, **T122**, **T131**, **T132**, **T141**, **T142**, **T311**, **T312**, **T321**, **T322**, **T331**, **T332**, **T341**, and **T342**.

2 The joints are located at X = **2.5 m**, and Y = **2.5 m** away from the vertical support joints **D101**, **D103**, **D201**, **D203**, **D301**, **D303**, **D401**, and **D403**. The *Fixities* for all added joints are **PINNED**.

3 Add members from newly added joints to jacket leg joints **101L**, **103L**, **201L**, **203L**, **301L**, **303L**, **401L**, and **403L**.

4 Define the *Member Property* as *OD* = **32.385 cm** and *Wall Thickness* = **1.27 cm**.

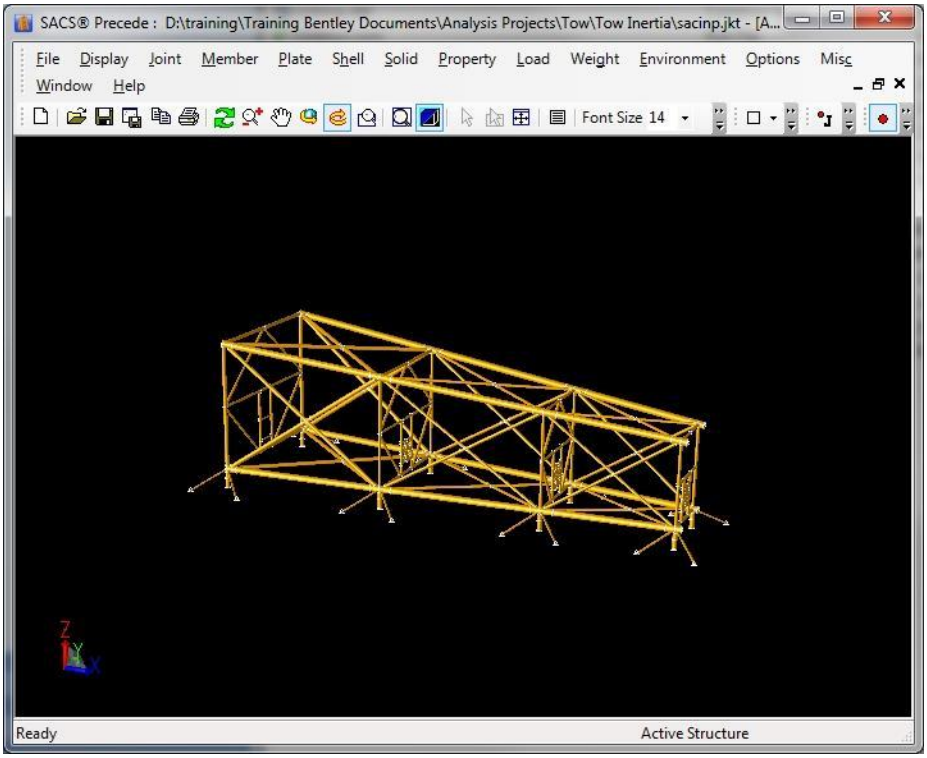
5 Set **Gap element** and select **No Load** from the list, see below.



6 Perform joint connection manual design to offset all tie-downs to the surfaces of jacket legs.

7 Add weight combination **MASS**, which includes weight groups of **ANOD**, **LPAD**, and **WKWY**.

8 Save the file **sacinp.jkt**.



➔ **Exercise: Revise the seastate input file to contain dead load only**

1 Open the seastate input file **seainp.loadout**.

2 Modify as shown below.

```
------------------------------------------------------------------------------
LDOPT        NF+Z1.0280007.849000    0.00    0.00GLOBMN
FILE B
LOAD
LOADCNDEAD
INCWGT   MASS
DEAD
DEAD       -Z                                M BML
END
------------------------------------------------------------------------------
```

3 Save the file as **seainp.dead**.

➔ **Exercise: Create tow input file TOWINP.TRAN**

1 Tow options need to be set to MN units and center of motion have the following coordinates

- X = **-75.0 m**
- Y = **3.5 m**
- Z = **4.5 m**

2 Weight combination group MASS is selected by using **INCWGT** line for inertia load generation.

3 Tow input file includes motions of **12.5 degree roll @ 10 seconds period**, **10 degree pitch @ 15 seconds period**, and the heave acceleration of **0.2 G**.

There are totally 8 load cases will be created using **MOTION** line, see below.

Load Case	Description
+R+H:	+ Roll + Heave
-R+H:	- Roll + Heave
+R-H:	+ Roll - Heave
-R-H:	- Roll - Heave
+P+H:	+ Pitch + Heave
-P+H:	- Pitch + Heave
+P-H:	+ Pitch - Heave
-P-H:	- Pitch – Heave

The gravity is only included in the lateral forces only. All eight load cases only contain the transportation inertial forces, no dead load included.

The tow input file generated shall looks like following:

```
------------------------------------------------------------------------
TRANSPORTATION ANALYSIS SAMPLE
TOWOPT  MN             OR       -75.00   3.50    4.5XYZ
INCWGT  MASS
MOTION+R+H 12.5 10.0                                0.2                 L
MOTION-R+H-12.5 10.0                                0.2                 L
MOTION+R-H 12.5 10.0                               -0.2                 L
MOTION-R-H-12.5 10.0                               -0.2                 L
MOTION+P+H           10.0 15.0                      0.2                 L
MOTION-P+H          -10.0 15.0                      0.2                 L
MOTION+P-H           10.0 15.0                     -0.2                 L
MOTION-P-H          -10.0 15.0                     -0.2                 L
END
------------------------------------------------------------------------
```

➔ **Exercise: Create a gap input file GAPINP.TRAN to override the tie-down members**

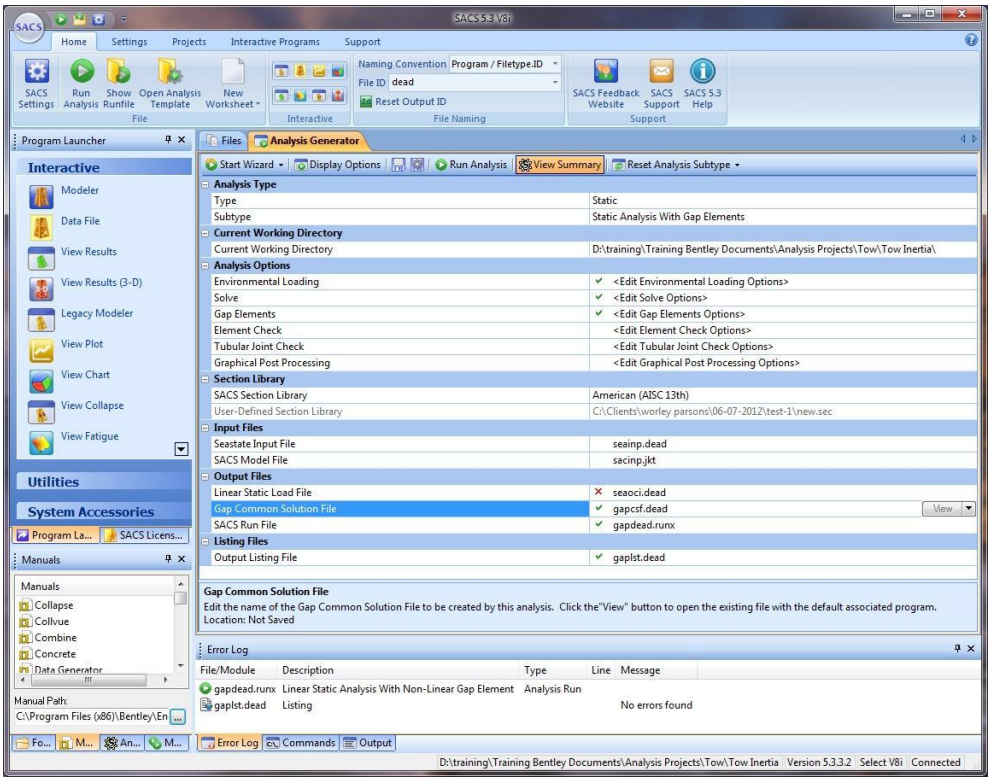
A gap input file, *gapinp.tran* is going to be used in tow analysis to override all no-load tie-down members to standard members.

Define members with *group ID* of **SEA** as standard members for the above 8 load cases.

```
--------------------------------------------------------------------------
GAPOPT          0     MN 600 0.00001           M
LCGAP +R+H INC ST GRP SEA
LCGAP -R+H INC ST GRP SEA
LCGAP +R-H INC ST GRP SEA
LCGAP -R-H INC ST GRP SEA
LCGAP +P+H INC ST GRP SEA
LCGAP -P+H INC ST GRP SEA
LCGAP +P-H INC ST GRP SEA
LCGAP -P-H INC ST GRP SEA
END
--------------------------------------------------------------------------
```

➔ **Exercise: Gap analysis for Dead Load only**

The analysis runfile is set-up as below.

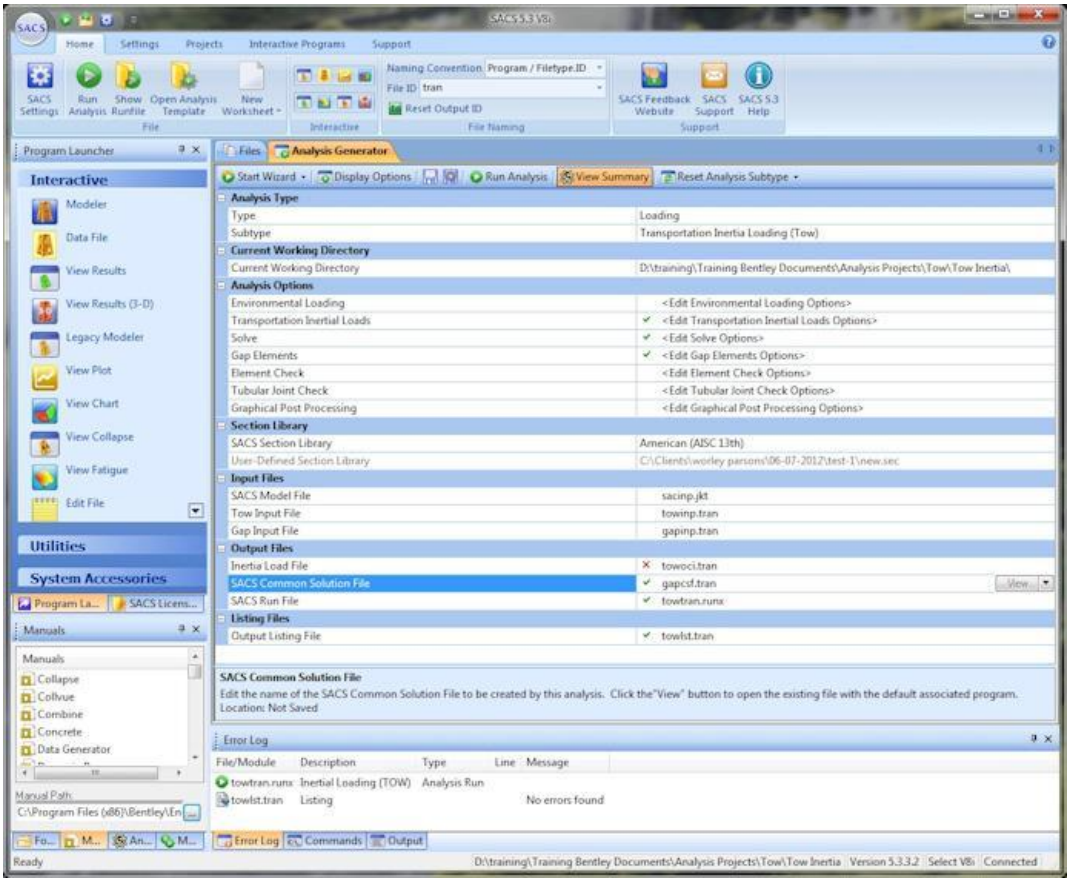


In this analysis only vertical support cans are effective. All tie-downs are No-Load GAP elements and they are not taking any load. Since all no-load gap elements were defined in the model file, there is no need to use a gap input file.

A common solution file *gapcsf.dead* is created in the analysis. It will be used in the combine analysis later.

➔ **Exercise: Tow analysis to create the transportation inertial loads**

The runfile is set-up as below.



A gap input file is used to redefine no-load tie-down members into standard members in order for all tie-down members to take the transportation forces.

Another common solution file *gapcsf.tran* is created in the analysis. This common solution file will be combined with the one created in the first analysis for post processing.

Gap analysis for dead load only:

FORCES AND MOMENTS SUMMARY
*** MOMENTS SUMMED ABOUT ORIGIN ***

LOAD CASE	FORCE (X)	FORCE (Y)	FORCE (Z)	MOMENT (X)	MOMENT (Y)	MOMENT (Z)
	KN			KN-M		
DEAD	0.000	0.000	6205.091	-0.671	255588.594	0.000

Transportation analysis forces and moments summary report:

FORCES AND MOMENTS SUMMARY
*** MOMENTS SUMMED ABOUT ORIGIN ***

LOAD CASE	FORCE (X)	FORCE (Y)	FORCE (Z)	MOMENT (X)	MOMENT (Y)	MOMENT (Z)
	KN			KN-M		
+R+H	0.000	1794.992	1284.690	-31173.586	52916.902	-75714.211
-R+H	0.000	-1794.999	903.194	31173.365	37202.453	75714.477
+R-H	0.000	1794.992	-1197.368	-31173.309	-49319.551	-75714.211
-R-H	0.000	-1794.999	-1578.864	31173.641	-65034.020	75714.484
+P+H	-1238.209	0.000	1801.884	-0.097	44298.785	-0.139
-P+H	1238.214	0.000	491.629	-0.151	50171.480	0.139
+P-H	-1238.209	0.000	-680.169	0.172	-57937.484	-0.141
-P-H	1238.214	0.000	-1990.429	0.115	-52065.004	0.140

➔ **Exercise: Combine the analysis results from the first two analyses**

1 Create a combine input file.

A combine input file is required to perform the combine analysis. In the combine input file the following info is included:

- It is defined that the DEAD load case from gap analysis for dead load only will be combined with each of the transportation load cases to create eight combined load cases C001 ~ C0008.
- The common solution file from gap analysis for dead load only is designated to be the primary file and the common solution file from transporation analysis is designated as the secondary file.

The combine input file is named **cominp.tow**. It is shown below.

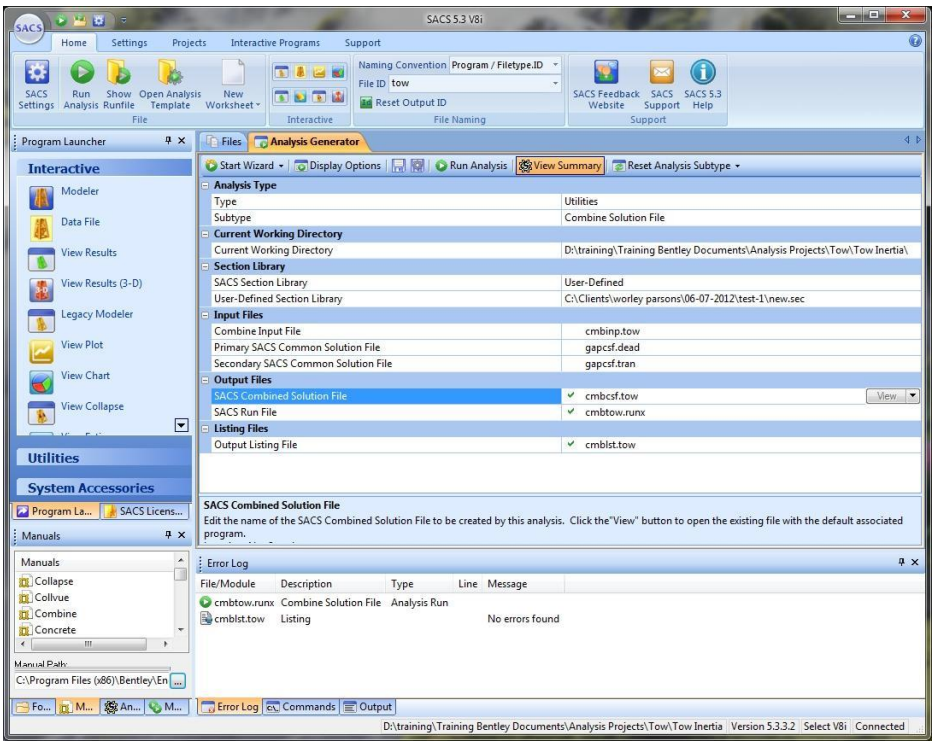
```
CMBOPT
LCOND C001   LIN     1.0
COMP PDEAD          1.0
COMP S+R+H          1.0
LCOND C002   LIN     1.0
COMP PDEAD          1.0
COMP S-R+H          1.0
LCOND C003   LIN     1.0
COMP PDEAD          1.0
COMP S+R-H          1.0
LCOND C004   LIN     1.0
COMP PDEAD          1.0
COMP S-R-H          1.0
```

```
LCOND C005    LIN     1.0
COMP  PDEAD           1.0
COMP  S+P+H           1.0
LCOND C006    LIN     1.0
COMP  PDEAD           1.0
COMP  S-P+H           1.0
LCOND C007    LIN     1.0
COMP  PDEAD           1.0
COMP  S+P-H           1.0
LCOND C008    LIN     1.0
COMP  PDEAD           1.0
COMP  S-P-H           1.0
END
```

---

2 Perform the combine analysis

- Combine input file: **cmbinp.tow**
- Primary SACS common solution file: **gapcsf.dead**
- Secondary SACS common solution file: **gapcsf.tran**
- Created SACS combined solution file: **cmbcsf.tow**



The combined analysis results are listed at the end of this document.

➔ **Exercise: Post processing**

1 Member code checks

- A post input file contains an *OPTIONS* line to define the member code check options and report options; an *LCSEL* line to specify the analysis load cases, an *UCPART* line to define the untiy check ranges, and an *AMOD* line to input the allowable stress modifers for each load case.

The post input file is shown below.

```
------------------------------------------------------------------------
OPTION        MN       SDUC    2 2    DC  C    PTPT      PT  PT
LCSEL ST         C001 C002 C003 C004 C005 C006 C007 C008
UCPART       0.5  0.8  0.8  1.0  1.0300.0
AMOD
AMOD   C001 1.333C002 1.333C003 1.333C004 1.333C005 1.333C006 1.333C007
1.333
AMOD   C008 1.333
END
------------------------------------------------------------------------
```

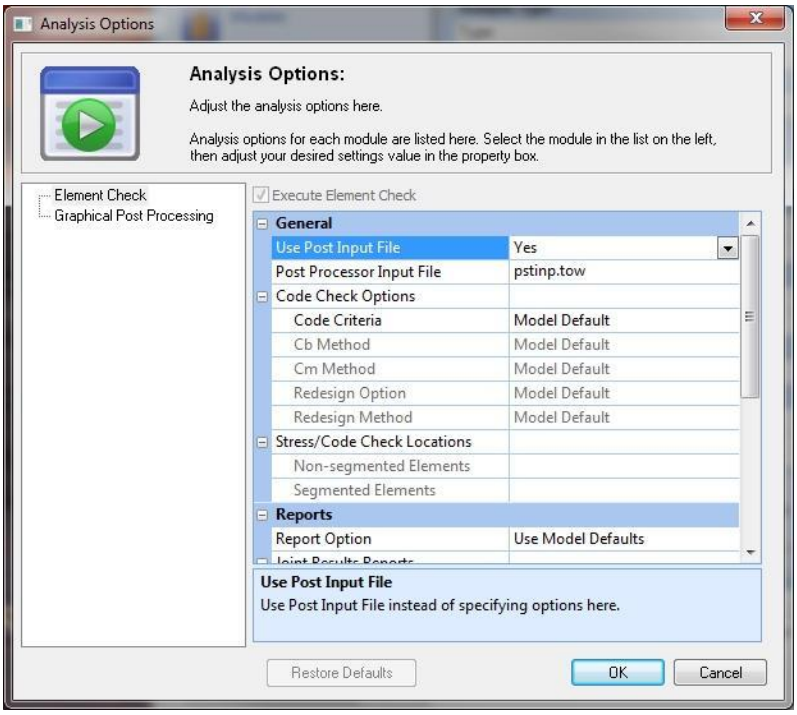
2 Set up the member code check runfile

- Analysis type: **Post Precessing**
- Analysis subtype: **Code Check**

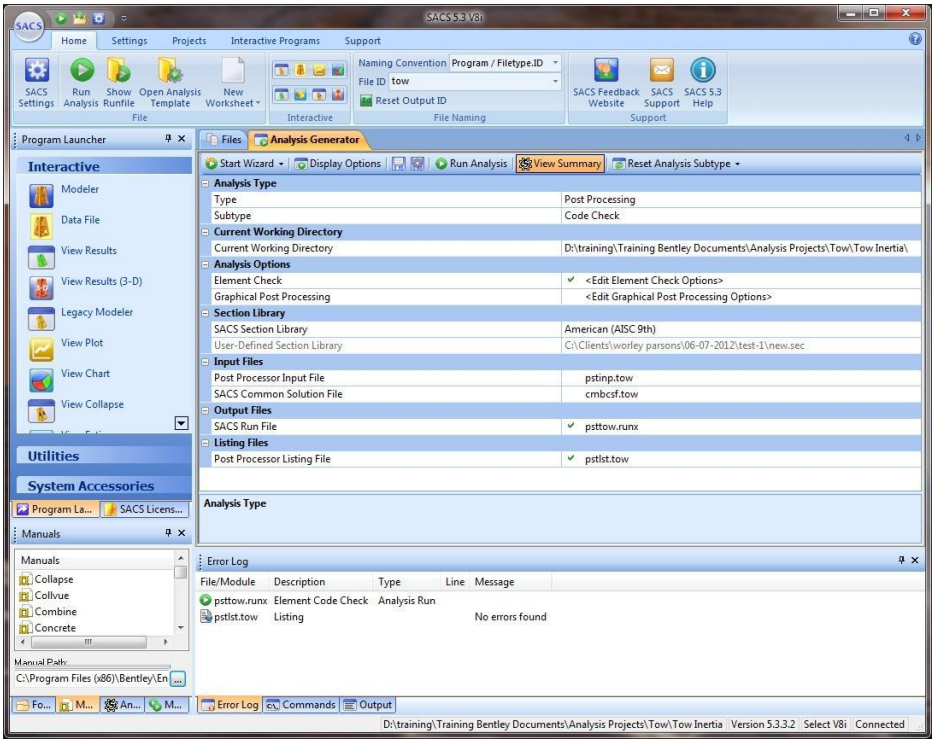
3 Click **<Edit Element Check Options>**.

4 Change *Use Post Input File* to **Yes**.

- SACS common solution file: **cmbcsf.tow**



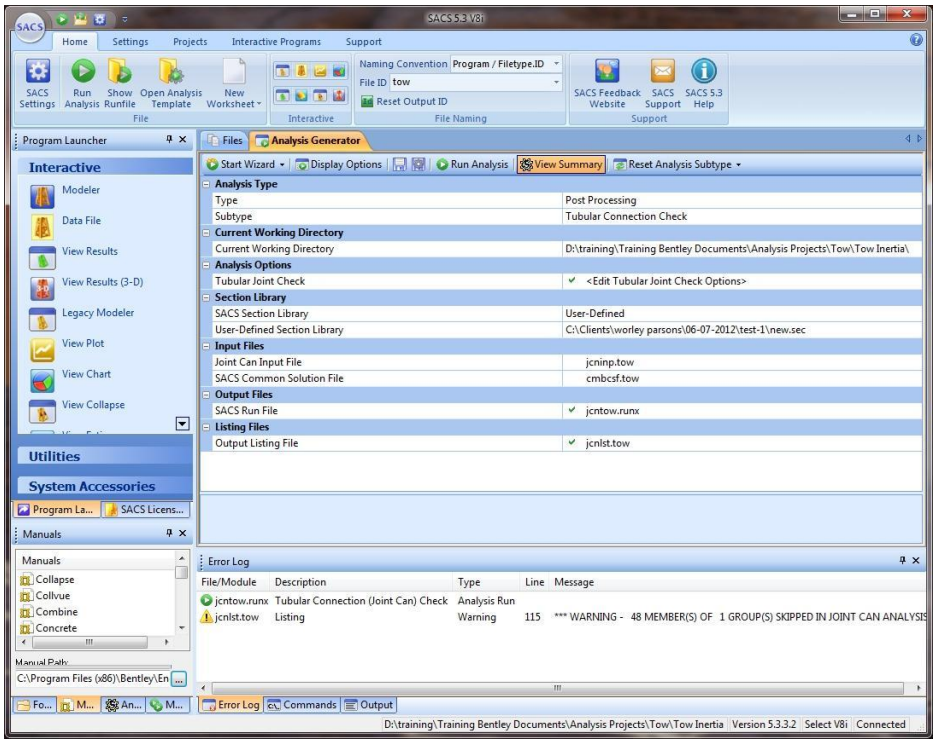
The following shows the runfile for member code checks.



Listing file **pstlst.tow** contains the analysis results.

5 Joint can code checks

- Copy the joint can input file from the Static Analysis directory to the current directory, and rename it to **jcninp.tow**.
- Set up the joint can check runfile:
  - Analysis type: **Post Processing**
  - Analysis subtype: **Tubular Connection Check**
  - Joint can input file: **jcninp.tow**
  - SACS Common solution file: **cmbcsf.tow**

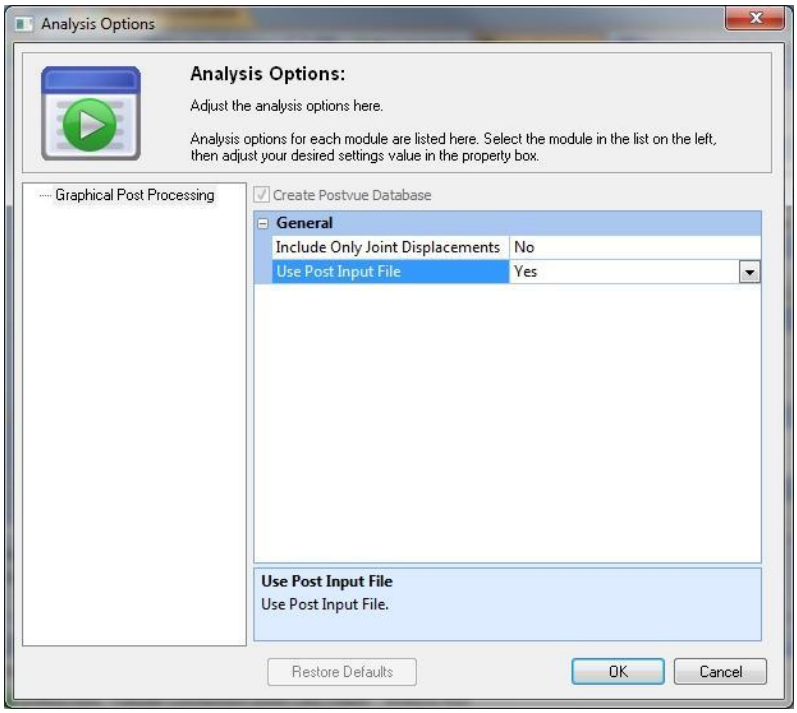


Listing file **jcnlst.tow** contains the analysis results.

6 Postvue database creation:

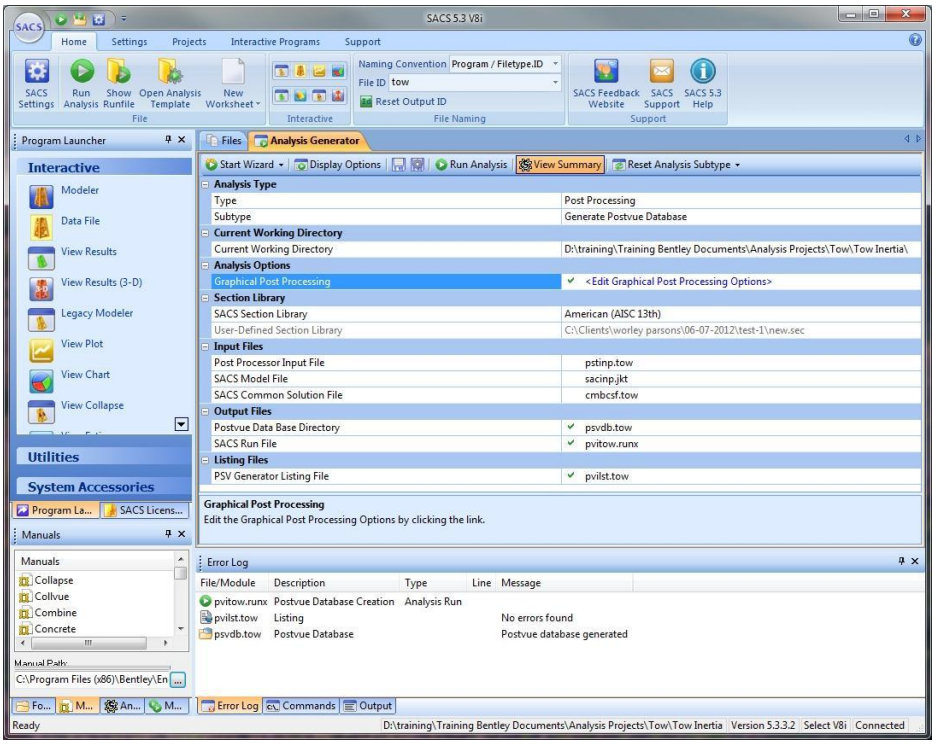
- Set up the runfile for Postvue database creation
  - Analysis type: **Post Processing**
  - Analysis subtype: **Generate Postvue Database**

7 Click **<Edit Graphical Post Processing Options>** and change *Use Post Input File* to **Yes**.



- Post processor input file: **pstinp.tow**
- SACS model file: **sacinp.jkt**

- SACS common solution file: **cmbcsf.tow**



Postvue database directory **psvdb.tow** is created and the listing file pvilst is empty if there is no error in the analysis.

Combined forces and moments summary reported in **cmblst.tow** file.

```
                                                  FORCES AND MOMENTS SUMMARY
                                              *** MOMENTS SUMMED ABOUT ORIGIN ***

                    ********************** KN **********************    ********************** KN-M
LOAD                FORCE(X)          FORCE(Y)          FORCE(Z)          MOMENT(X)          MOMEN
CASE

C001                   0.000          1794.992          7489.782         -31174.248          308505
C002                   0.000         -1794.999          7108.286          31172.697          292791
C003                   0.000          1794.992          5007.724         -31173.973          206269
```

Total joint reactions reported in pstlst.tow:

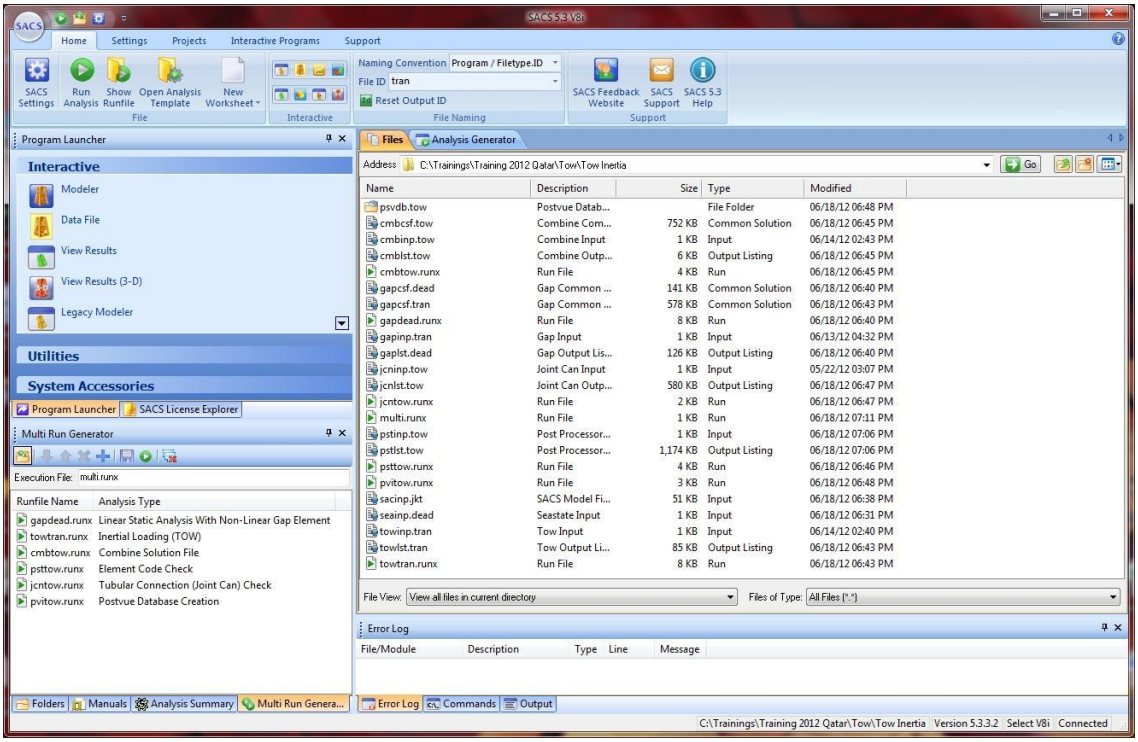
```
                        SACS-IV SYSTEM    FIXED JOINTS REACTION FORCES AND MOMENTS SUMMARY
                                              *** MOMENTS SUMMED ABOUT ORIGIN ***

                    ********************** KN **********************    ********************** KN-M **********************
LOAD                FORCE(X)          FORCE(Y)          FORCE(Z)          MOMENT(X)          MOMENT(Y)          MOMENT(Z)
CASE

C001                   0.000          1794.991          7489.783         -31174.252         308505.594         -75714.203
C002                   0.000         -1794.998          7108.286          31172.697         292791.188          75714.469
C003                   0.000          1794.992          5007.723         -31173.980         206269.094         -75714.219
C004                   0.000         -1794.999          4626.229          31172.969         190554.625          75714.477
C005               -1238.209             0.000          8006.975             -0.769         299887.469             -0.137
C006                1238.214             0.000          6696.721             -0.823         305760.094              0.138
C007               -1238.209             0.000          5524.922             -0.499         197651.188             -0.139
C008                1238.214             0.000          4214.663             -0.555         203523.625              0.141
```

➔ **Exercise: Multiple runs**

1 In this transportation analysis there are total six runfiles created:

- Gap dead load analysis: **gapdead.runx**
- Tow inertial loads analysis: **towtran.runx**
- Combine analysis: **comtow.runx**
- Post processing – member code checks: **psttow.runx**
- Post processing – joint can code checks: **jcntow.runx**
- Post processing – postvue database generation: **pvitow.runx**

SACS is able to make a multiply run including all analysis runfiles in a multi-runfile, see below.



2 Click the icon **Run Multi Analyses** to run all six analyses one by one.