

Tutorial Problem

1. Model description:

The frame is 10' x 20', the upright columns are 10" x 10" x 1/4" box sections and the horizontal beam is an AISC W18x40 wide flange. Three load cases will be considered, Load case 1 is a uniform dead load of -1.0 kips/ft along the horizontal beam; Load case 2 is a 16 kips concentrated wind load; Load case 3 is a load combination of load case 1 and 2, each at 100% and factored at 133%.

2. Create Project and major task "Structural Modeling":

Launch SACS Executive;

Under **Project/Task** menu select **Add project**

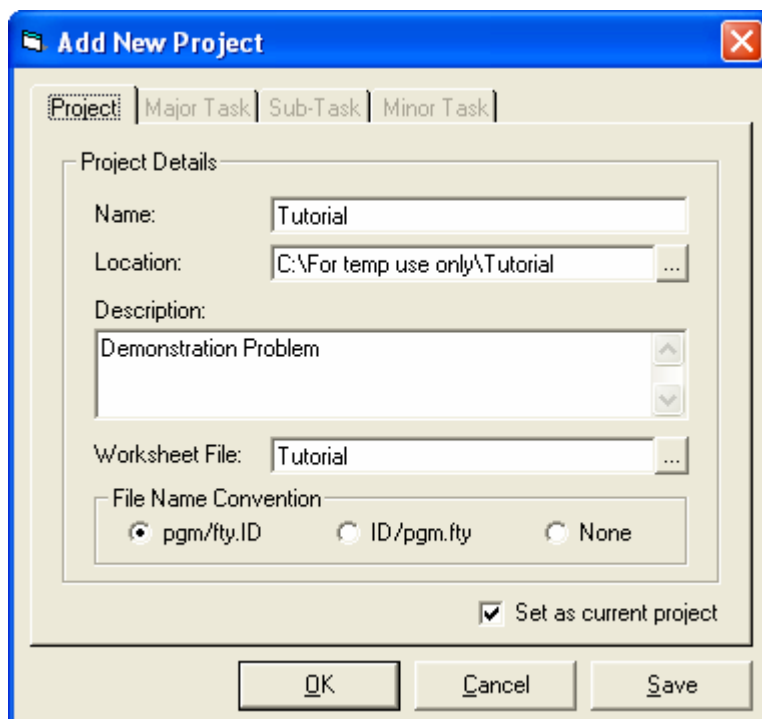
Type in Name "**Tutorial**"

Type in Location "**C:\...\Tutorial**"

Type in Description "**Demonstration Problem**"

Type in Worksheet file "**Tutorial**" then click **OK**

Accept and create the new directory (Refer to following "Add New Project" figure)



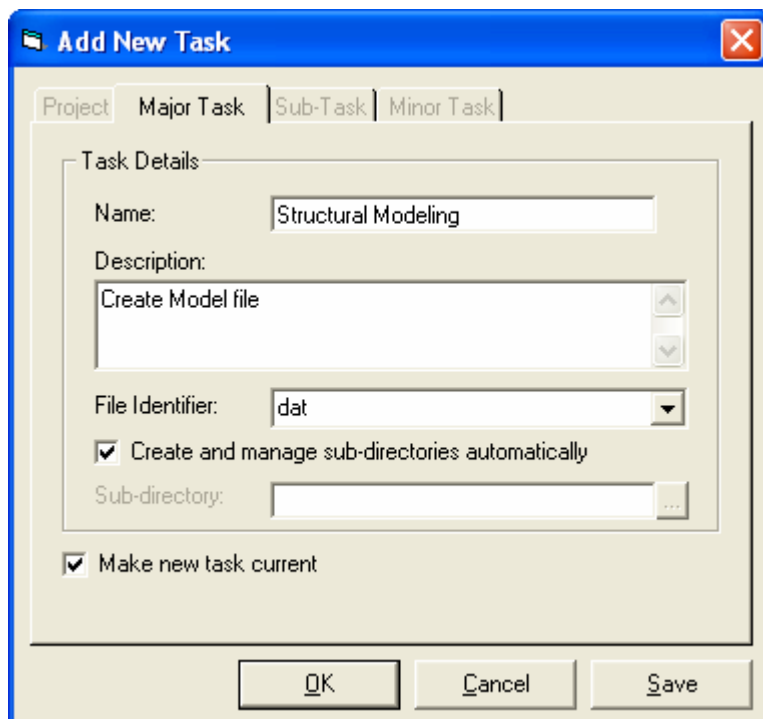
Under **Project/Task** menu select **Add major task**

Type in Name **Structural Modeling**

Type in description **Create Model File**

Type in File Identifier **dat**

Then click OK (Refer to following "Add New Task" figure)



Add New Task

Project | **Major Task** | Sub-Task | Minor Task

Task Details

Name: Structural Modeling

Description: Create Model file

File Identifier: dat

☒ Create and manage sub-directories automatically

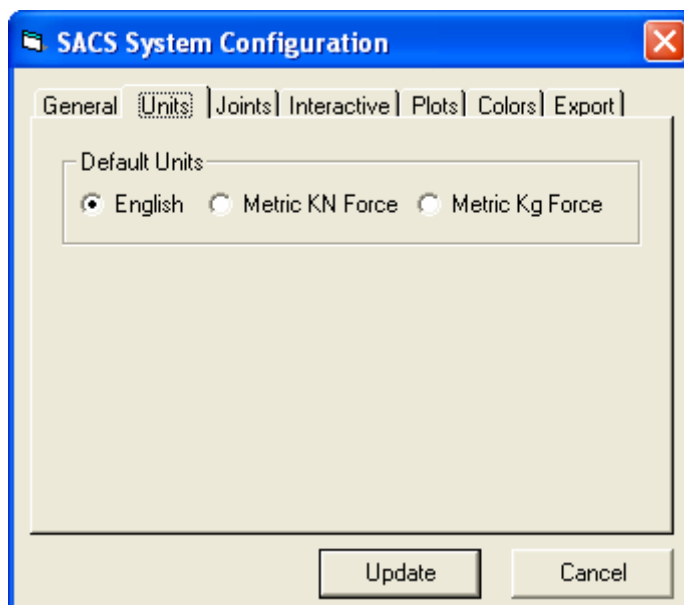
Sub-directory:

☒ Make new task current

OK Cancel Save

3. Model generation using PRECEDE:

Make sure the unit is correctly set to English. Under **Settings > SACS System Configuration > Units**, select English unit if necessary. (Refer to “**SACS System Configuration**” figure)



SACS System Configuration

General | **Units** | Joints | Interactive | Plots | Colors | Export

Default Units

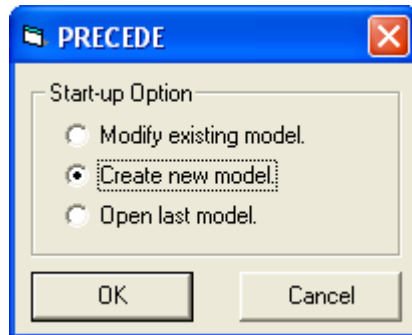
☒ English ☐ Metric KN Force ☐ Metric Kg Force

Update Cancel

a) **Launch PRECEDE**

Click **Modeling > Precede**

Select **“Created new model”** then **OK** (Refer to **“PRECEDE”** Figure)



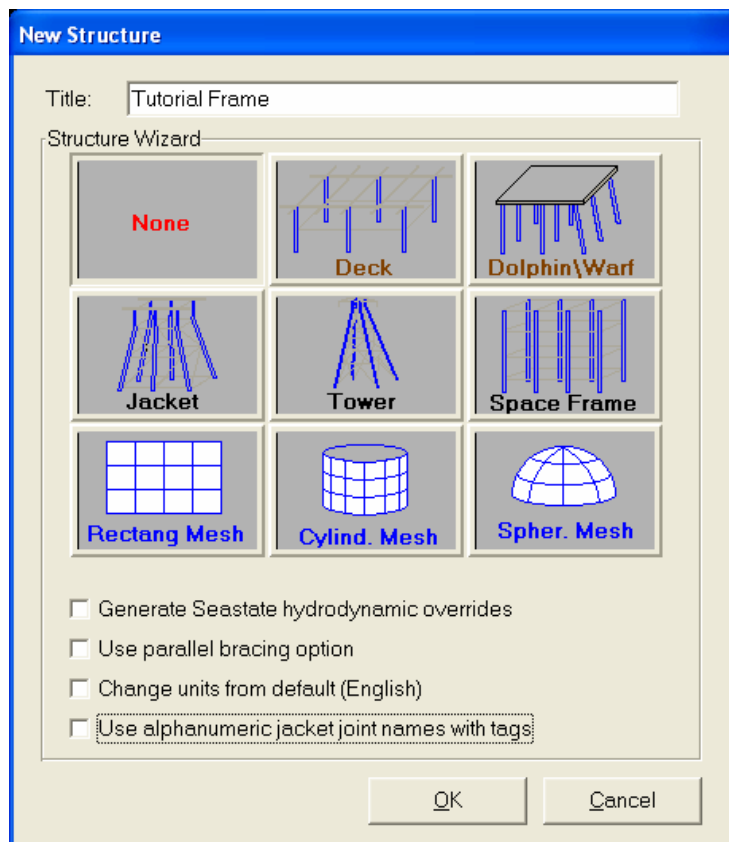
For new structure select **“None”**;

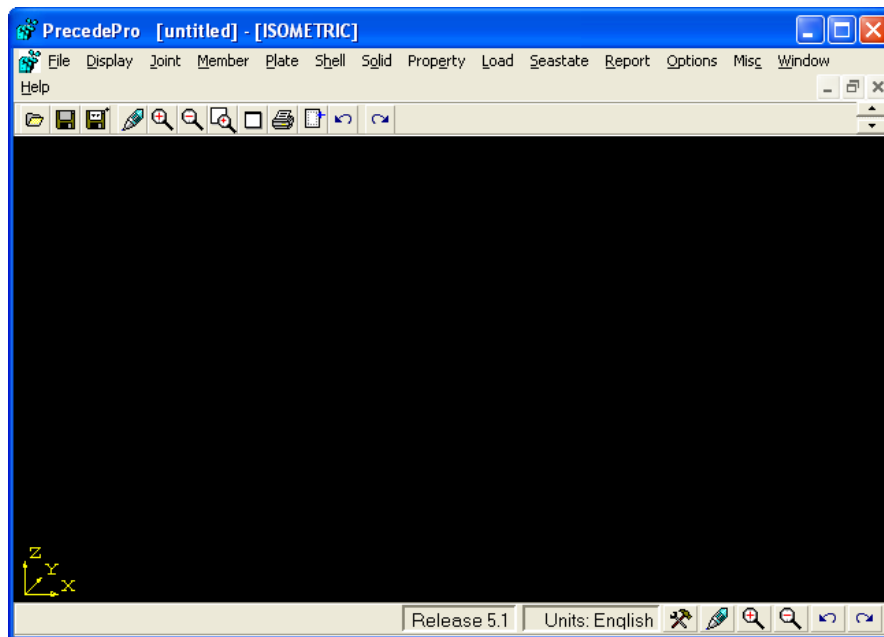
Type in Title **“Tutorial Frame”**

Deselect **“Use alphanumeric jacket joint names with tags”**

Then click **OK** (Refer to **“New Structure”** Figure)

PRECEDE program will be launched (Refer to **“PrecedePro”** figure)



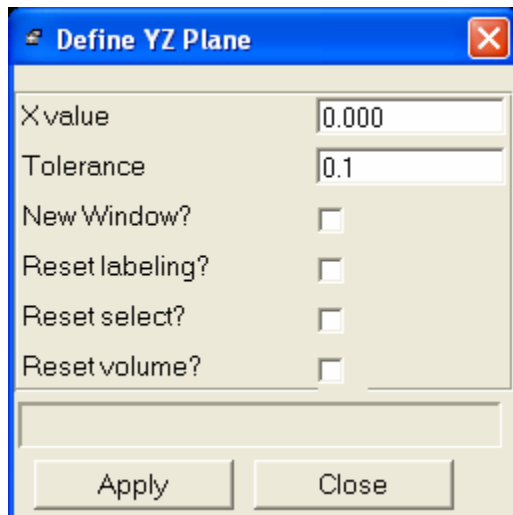


b) Create joints

Add joint #1 with absolute coordinates of 0, 0, 0

Add joint #2 with absolute coordinates of 0, 0, 10.0

Use menu command **Display > Plane > YZ Plane** Change current 3d view to YZ plane, click any one of the two joints for X value coordinates. (Refer to “Define YZ Plane” Figure)



Use **Display > Zoom Box > Translate/Rotate > General** command

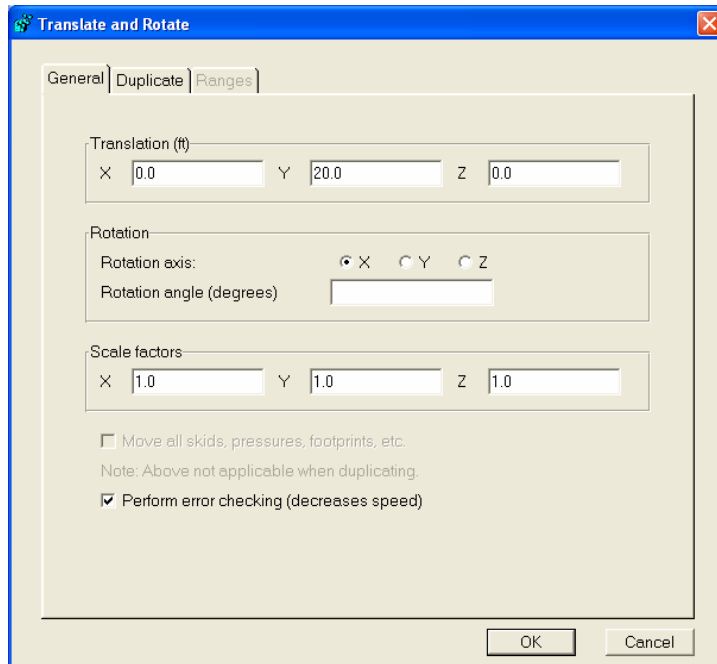
Select by drawing a window includes joint #1 and #2,

Type in Translation Y “**20.0**” (Refer to “Translate and Rotate – General” Figure)

Select “**Duplicate**” then “**Duplicate existing joints**”

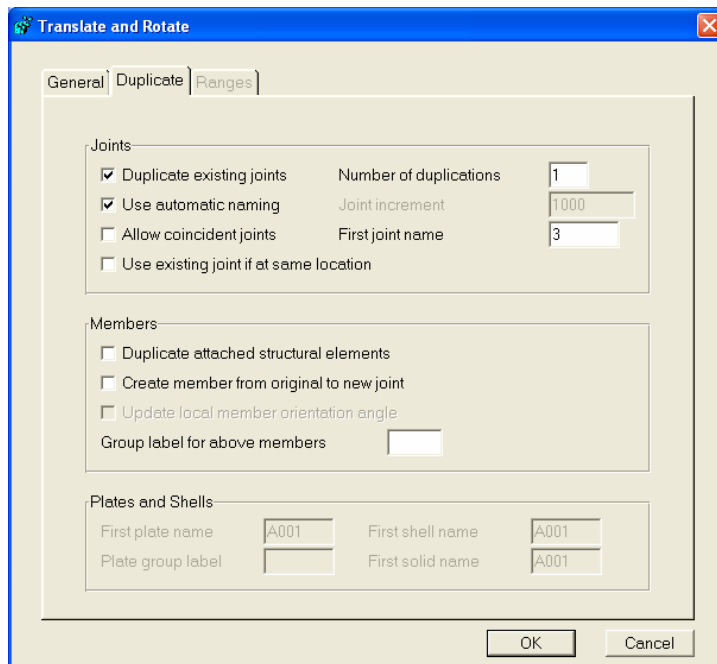
Type in Number of duplications “1” (Refer to “**Translate and Rotate – Duplicate**” Figure)

Click OK and created joints #3 and #4



The 'Translate and Rotate' dialog box is shown with the 'General' tab selected. It contains the following fields and options:

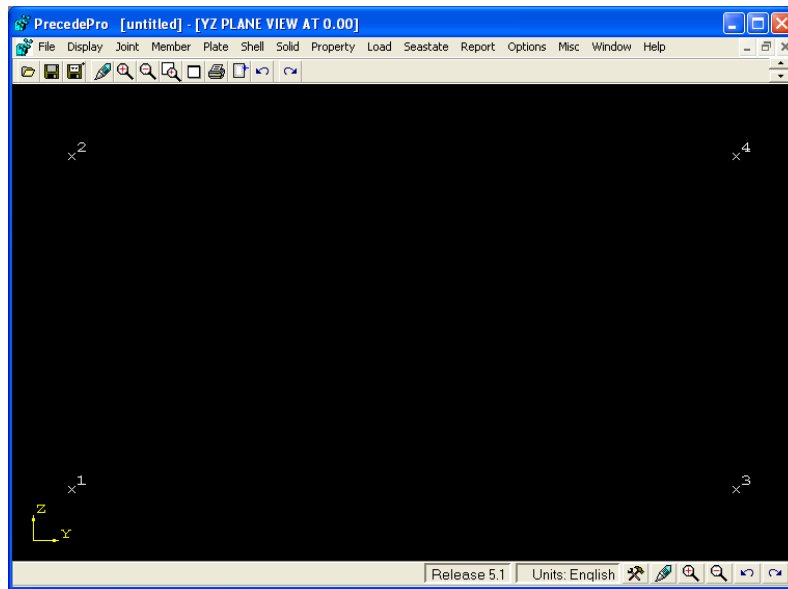
- Translation (ft):** X: 0.0, Y: 20.0, Z: 0.0
- Rotation:** Rotation axis: X (selected), Y, Z. Rotation angle (degrees):
- Scale factors:** X: 1.0, Y: 1.0, Z: 1.0
- ☐ Move all skids, pressures, footprints, etc.
- Note: Above not applicable when duplicating.
- ☒ Perform error checking (decreases speed)
- Buttons: OK, Cancel



The 'Translate and Rotate' dialog box is shown with the 'Duplicate' tab selected. It contains the following fields and options:

- Joints:**
 - ☒ Duplicate existing joints. Number of duplications: 1
 - ☒ Use automatic naming. Joint increment: 1000
 - ☐ Allow coincident joints. First joint name: 3
 - ☐ Use existing joint if at same location
- Members:**
 - ☐ Duplicate attached structural elements
 - ☐ Create member from original to new joint
 - ☐ Update local member orientation angle
 - Group label for above members:
- Plates and Shells:**
 - First plate name: A001. First shell name: A001
 - Plate group label: . First solid name: A001
- Buttons: OK, Cancel

Use **Display > Labeling > Joint** to display joint names. We have created four corner joints of the frame. (Refer to “**PrecedePro [untitled]**” Figure)



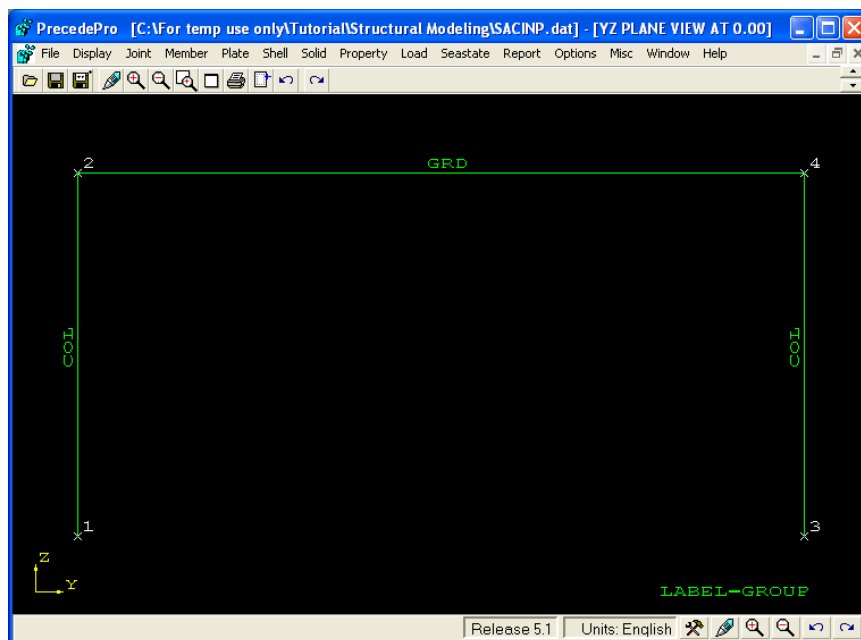
c) **Save and register the file in the project by accepting all the default names and settings, the saved file name shall be SACINP.DAT**

d) **Add frame members**

Add member 1 to 2 and 3 to 4 as group “**COL**”;

Add member 2 to 4 as group “**GRD**”

Use **Display > Labeling > Members** to display members groups. (Refer to “Precadepro [C:\...\Tutorial\Structural Modeling\SACINP.DAT” figure)



e) **Define member properties**

Select **Property > Member Group** and Choose “COL” then “Define”

Select Group type as “General”

Type in Section label “Box10” (Refer to “Define Member Group COL” Figure)

Define Member Group COL Segment 1 of 1

General | Post Processing

Group type: General

Section label: BOX10

E modulus (x1000) (ksi): 29.000

G modulus (x1000) (ksi): 11.600

Yield strength (ksi): 36.000

Density (lb/cu ft): 490.000

Segment length (ft):

☐ Flooded member

☐ Tapered section ☒ Beginning ☐ End

☐ Gap element ☒ Tension ☐ Compression ☐ No load ☐ Friction

Copy group label: Copy

AddSeg OK Cancel

Choose “GRD” then “Define”

Select Group type as “General”

Browse in section label to find “W18X40” (Refer to “Define Member Group GRD” Figure)

Define Member Group GRD Segment 1 of 1

General | Post Processing

Group type: General

Section label: W18X40

E modulus (x1000) (ksi): 29.000

G modulus (x1000) (ksi): 11.600

Yield strength (ksi): 36.000

Density (lb/cu ft): 490.000

Segment length (ft):

☐ Flooded member

☐ Tapered section ☒ Beginning ☐ End

☐ Gap element ☒ Tension ☐ Compression ☐ No load ☐ Friction

Copy group label: Copy

AddSeg OK Cancel

Select “**Close**” to quit member group definition window

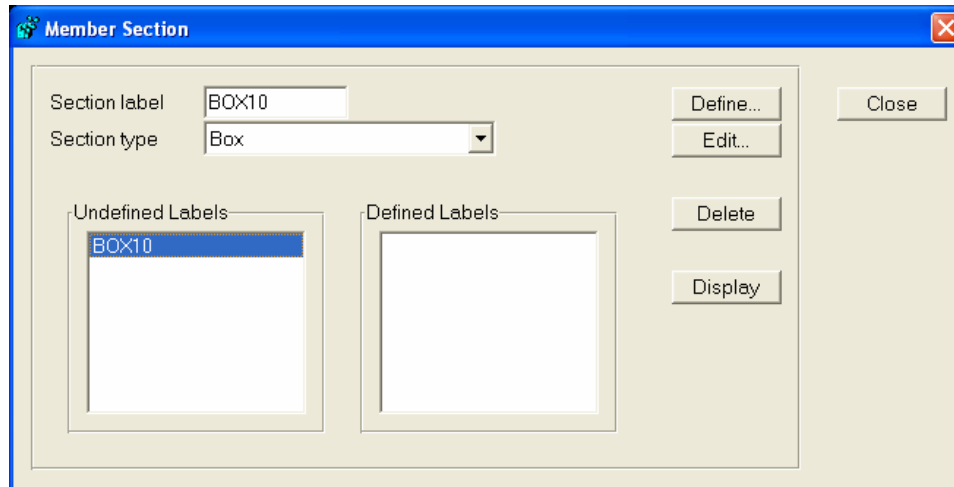
f) Define member sections

Select **Property > Member Section**

Choose “**Box10**”

Select Section Type as “**Box**”

Then “**Define**” (Refer to “**Member Section**” Figure)

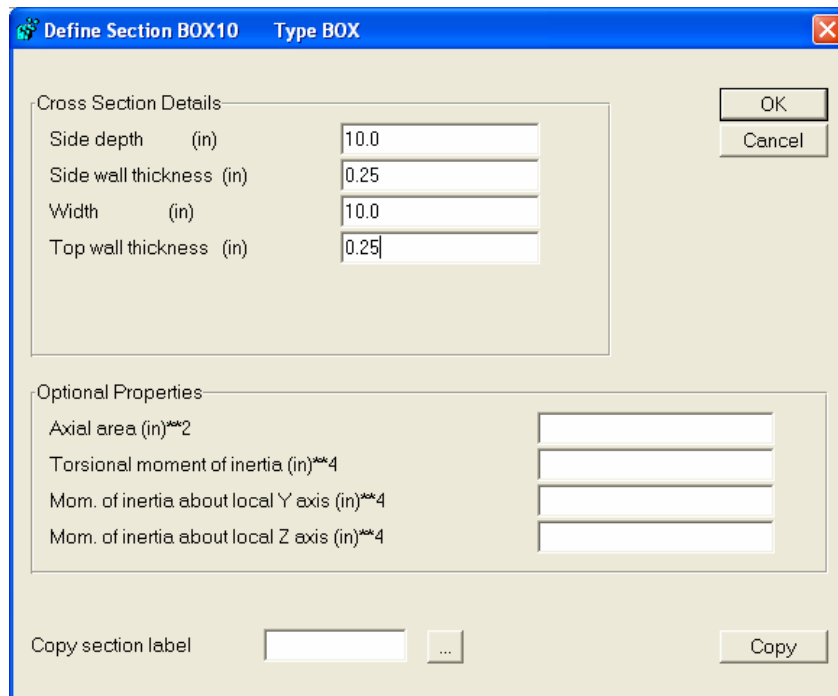


Type in Side depth “**10.0**”

Type in Side wall thickness “**0.25**”

Type in Width “**10.0**”

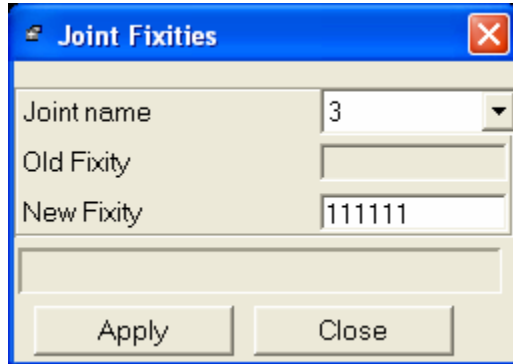
Type in Top wall Thickness “**0.25**” (Refer to “**Define Section BOX10**” Figure)



Then **OK** and “**Close**” to quit

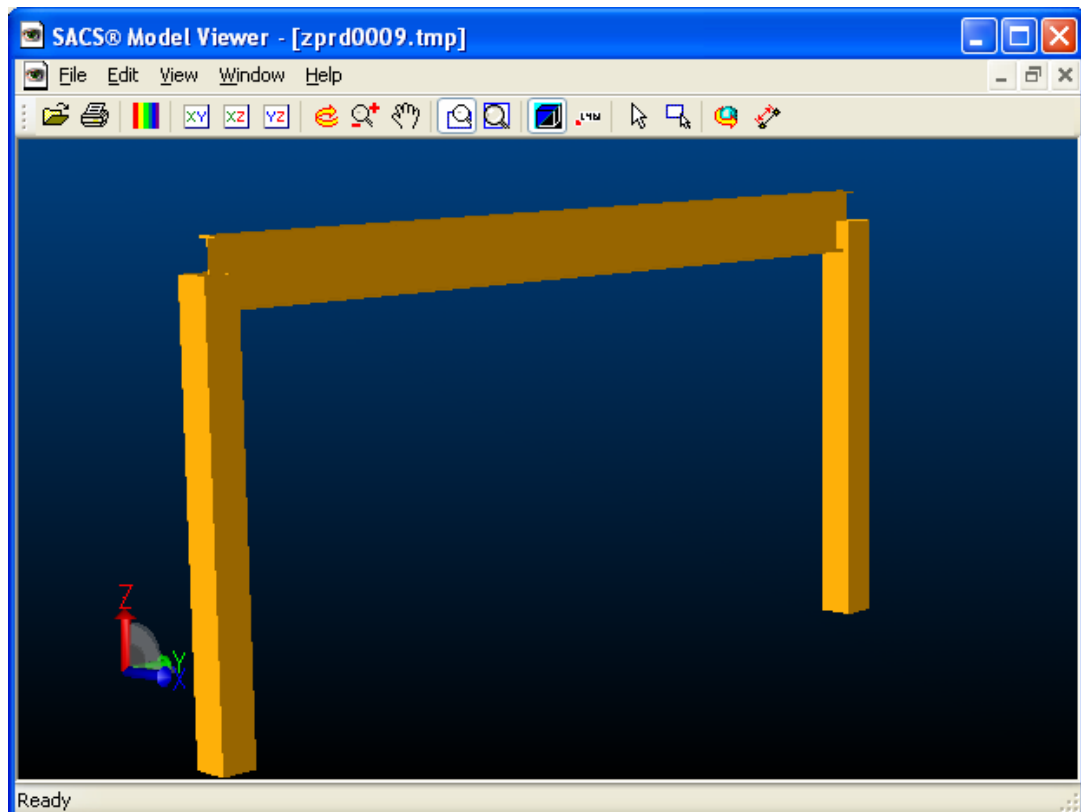
g) Define joint fixities

Select “Joint” > “Fixities” and Choose Joint #1 and #3 (Hold Ctrl key to perform multiple joint selection),
Type in New Fixity “111111” then “Close” to quit. (Refer to “**Joint Fixities**” Figure)



h) Save and view 3D in Model viewer

Use **Display > Model Viewer** to display 3d model.



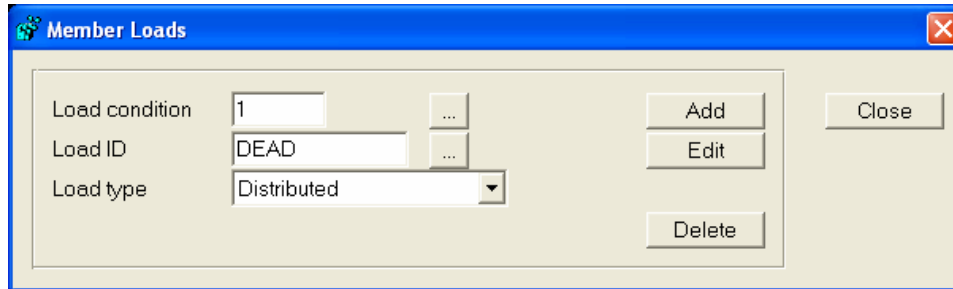
i) **Add load case #1**

Select **Load > Members** then Choose member 2-4 and click “**Apply**”

Type in Load Condition “**1**”

Type in Load ID “**Dead**”

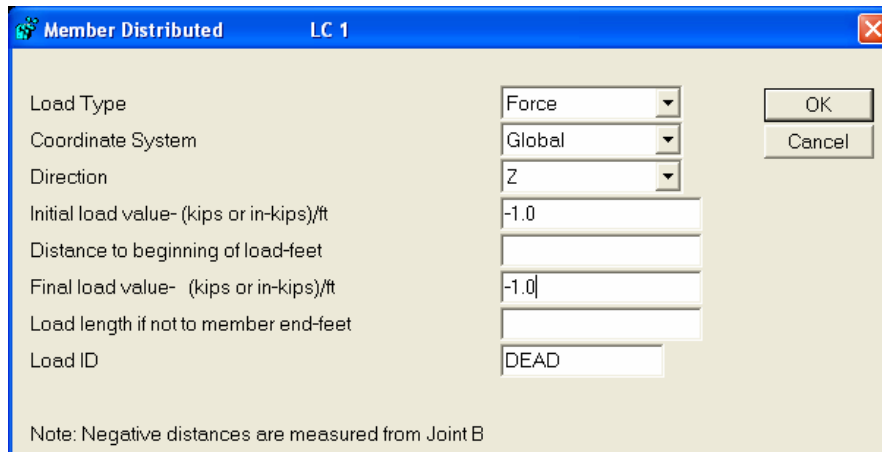
Select Load type as “**Distributed**” then Click “**Add**” (Refer to “**Member Loads**” Figure)



The "Member Loads" dialog box has a blue title bar with a close button. It contains three input fields: "Load condition" with the value "1", "Load ID" with the value "DEAD", and "Load type" with a dropdown menu showing "Distributed". To the right of these fields are four buttons: "Add", "Edit", "Delete", and "Close".

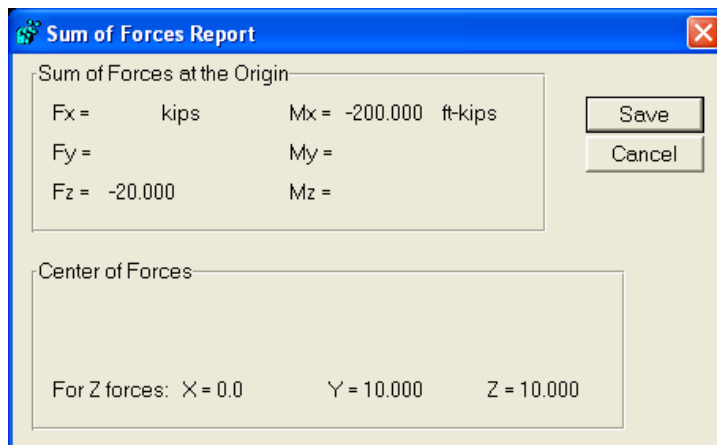
Type in Initial load value “-1.0”

Type in final load value “-1.0” then click OK (Refer to “**Member Distributed**” Figure)



The "Member Distributed" dialog box has a blue title bar with the text "Member Distributed" and "LC 1", and a close button. It contains several input fields: "Load Type" (Force), "Coordinate System" (Global), "Direction" (Z), "Initial load value- (kips or in-kips)/ft" (-1.0), "Distance to beginning of load-feet" (empty), "Final load value- (kips or in-kips)/ft" (-1.0), "Load length if not to member end-feet" (empty), and "Load ID" (DEAD). To the right are "OK" and "Cancel" buttons. A note at the bottom states: "Note: Negative distances are measured from Joint B".

Review the force summary report and Click “**Save**”, the load will display in the graphics window also. (Refer to “**Sum of Forces Report**” Figure)



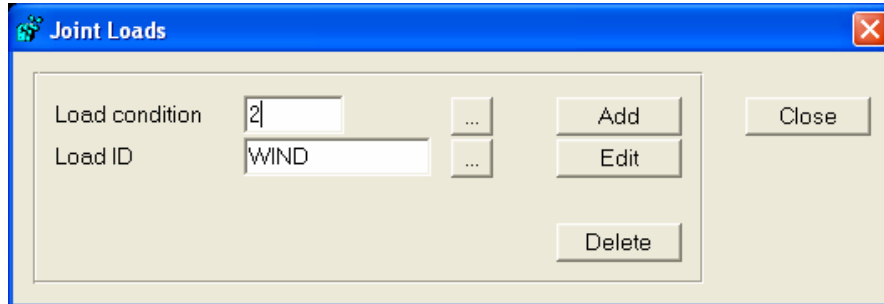
The "Sum of Forces Report" dialog box has a blue title bar with a close button. It contains two sections. The first section, "Sum of Forces at the Origin", shows: Fx = kips, Mx = -200.000 ft-kips, Fy = , My = , Fz = -20.000, and Mz = . The second section, "Center of Forces", shows: For Z forces: X = 0.0, Y = 10.000, Z = 10.000. To the right of the first section are "Save" and "Cancel" buttons.

j) **Add load case #2**

Select **Load > Joints** then Choose joint #2 and click **“Apply”**

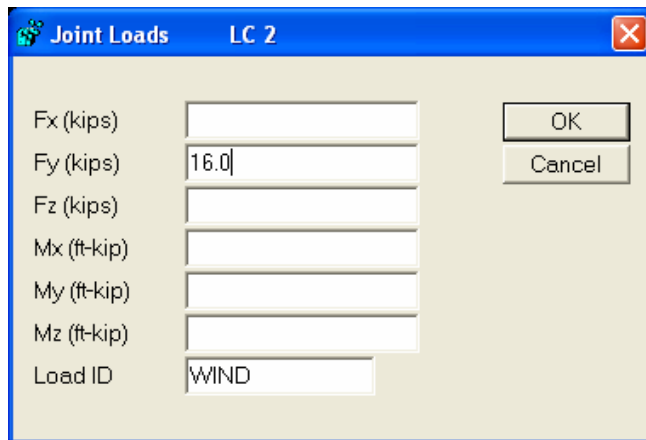
Type in Load Condition **“2”**

Type in Load ID **“WIND”** then **“Add”** (Refer to **“Joint Loads”** Figure)



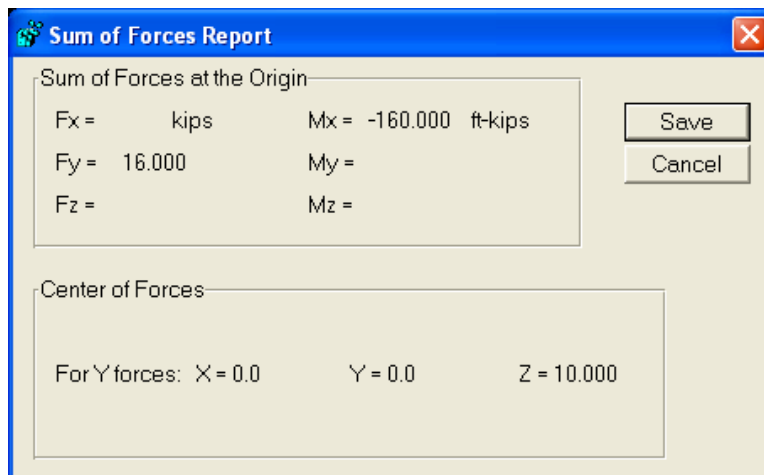
The 'Joint Loads' dialog box has a blue title bar with a green icon and a close button. It contains two input fields: 'Load condition' with the value '2' and 'Load ID' with the value 'WIND'. Each field has a dropdown arrow to its right. To the right of the input fields are three buttons: 'Add', 'Edit', and 'Delete'. A 'Close' button is located to the right of the 'Add' button.

Type in Fy **“16.0”** then OK (Refer to **“Joint Loads LC2”** Figure)



The 'Joint Loads LC 2' dialog box has a blue title bar with a green icon, the text 'LC 2', and a close button. It contains six input fields for force components: Fx (kips), Fy (kips), Fz (kips), Mx (ft-kip), My (ft-kip), and Mz (ft-kip). The Fy field contains the value '16.0'. Below these is a 'Load ID' field containing 'WIND'. To the right of the input fields are 'OK' and 'Cancel' buttons.

Review force summary report and **“Save”**, note the force also shown in the graphics window. (Refer to **“Sum of Forces Report”** Figure)



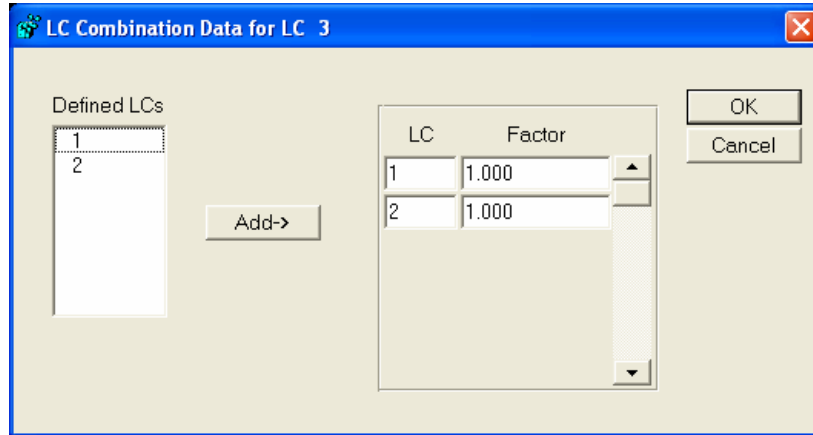
The 'Sum of Forces Report' dialog box has a blue title bar with a green icon and a close button. It contains two sections. The first section, 'Sum of Forces at the Origin', shows Fx = kips, Fy = 16.000, Fz = kips, Mx = -160.000 ft-kips, My = kips, and Mz = kips. The second section, 'Center of Forces', shows 'For Y forces: X = 0.0, Y = 0.0, Z = 10.000'. To the right of the first section are 'Save' and 'Cancel' buttons.

k) **Add load case #3**

Select **Load > Combine Load Conditions**

Type in LC combination label “**3**” then “**Define**”

Choose Load case 1 and 2 then “**OK**” and “**Close**” to quit (Refer to “**LC Combination Data for LC 3**” Figure)

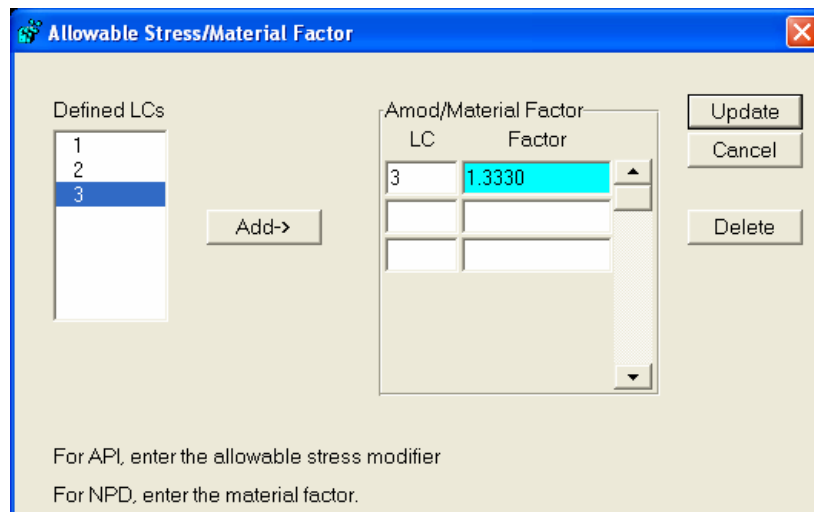


l) **Save**

m) **Define allowable stress modifier**

Select **Options > Allowable Stress/Mat Factor**

Choose load case 3 and type in factor as “**1.333**” then “**Update**” to quit (Refer to “**Allowable Stress/Material Factor**” Figure)



Save, Exit PRECEDE program.

4. Use Datagen to make modifications to model file SACINP.DAT:

Choose **Modeling > DataGen**, select **Edit existing data file** and select **SACINP.DAT**.

Change options line: Double click the **OPTIONS** line, make necessary changes as

- General Window: select **APUC** for code check option – Tubular API 21st edition and others AISC 9th edition; (Refer to “**SACS Options – General**” Figure)

SACS Options

General | Report

Working Units: ☒ EN ☐ ME ☐ MN

☐ Include P-Delta Effects

Code Check Option: **APUC**

☐ Super Element Input ☐ Super Element Creation

☒ Shear Deformation Included ☐ Member Release Override

Stress Option: ☒ Dflt ☐ JT ☐ JO

☐ End Moment Cb Calculation ☐ Exclude Moment Magnification

☐ DKT Thin Plate Theory ☐ Plate Element Check

LRFD PHI Factors: **C**

☐ Include Joint Flexibility

< Prev Next > OK Cancel Help

- Report Window: Choose appropriate print options as desired (Refer to “**SACS Options – Report**” Figure)

SACS Options

General | Report

Echo Input: ☒ NO ☐ PT ☐ NL

☐ Interpretive Input Data Report ☒ Joint Deflection Report

Element Detail Report: ☒ None ☐ All ☐ Selc

☒ Member Internal Loads Report ☒ Joint Reactions Report

☐ Stress at Maximum Unity Check Report ☐ Member Combined and Shear UC Report

☐ Plate, Shell and Member UC Report

No. C.C Parts for non-segmented members: **2**

No. of C.C. Parts for segmented members: **1**

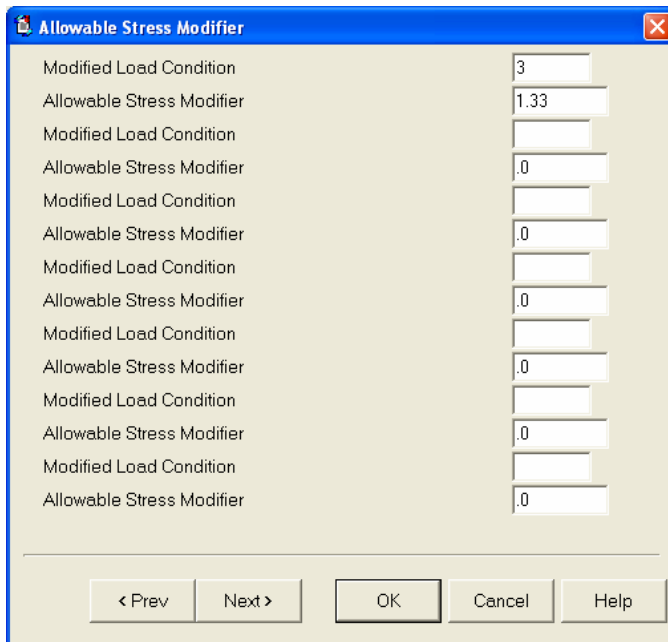
Supplemental File Option: **NO**

Special Element Report: ☒ DFLT ☐ PRNT ☐ SKIP

☒ Unity Check Range Report

< Prev Next > OK Cancel Help

Double Click **AMOD** line and Change allowable stress modifier from **1.333** to **1.33**
(Refer to “**Allowable Stress Modifier**” Figure)



The "Allowable Stress Modifier" dialog box contains a list of 14 rows. Each row has two columns: "Modified Load Condition" and "Allowable Stress Modifier". The first row has values "3" and "1.33". The remaining 13 rows have values ".0" and ".0". At the bottom, there are five buttons: "< Prev", "Next >", "OK", "Cancel", and "Help".

Save file and **exit** from DataGen program, the model file now is ready to run.

5. Create Major Task “Static Analysis”:

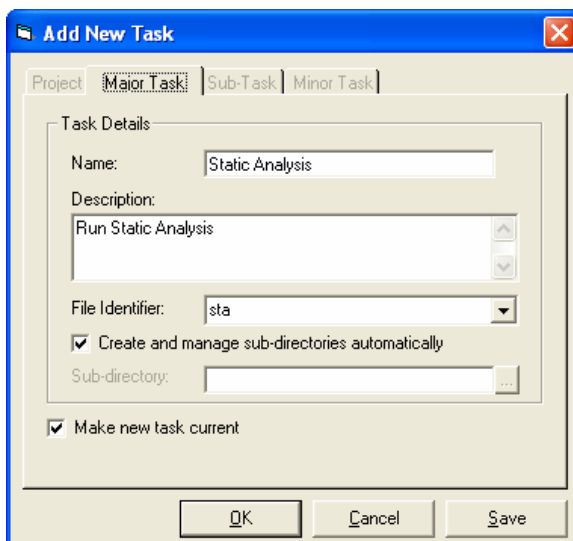
Use **Project/Task > Add major task** again

Type in Name “**Static Analysis**”

Type in description “**Run Static Analysis**”

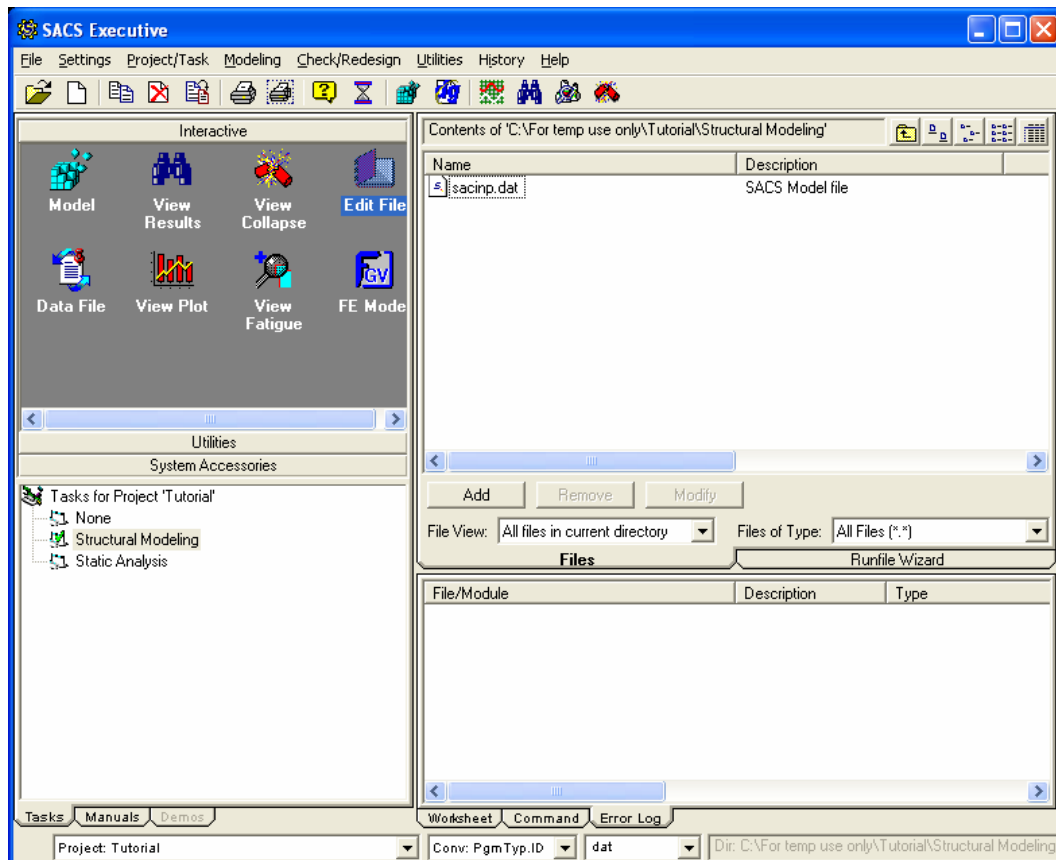
Type in File Identifier “**sta**”

Then click OK (Refer to “**Add New Task**” Figure)

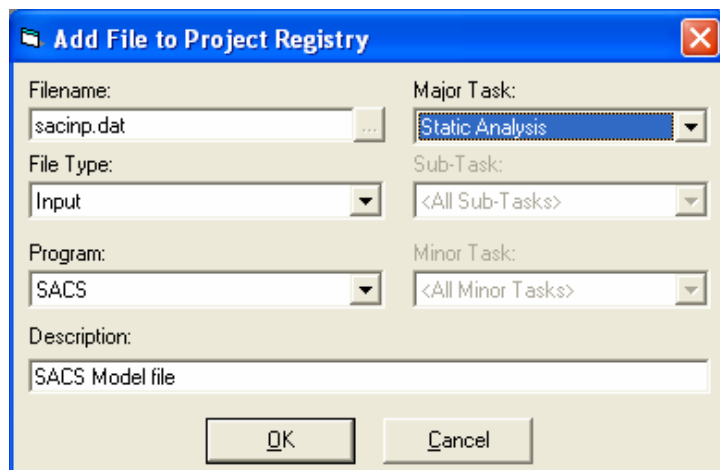


The "Add New Task" dialog box has four tabs: "Project", "Major Task", "Sub-Task", and "Minor Task". The "Major Task" tab is selected. Under "Task Details", there are four fields: "Name:" with the value "Static Analysis", "Description:" with the value "Run Static Analysis", "File Identifier:" with the value "sta", and "Sub-directory:" with a text box and a browse button. There are two checkboxes: "Create and manage sub-directories automatically" (checked) and "Make new task current" (checked). At the bottom, there are three buttons: "OK", "Cancel", and "Save".

Choose **Structural Modeling** task from **Tasks for Project 'Tutorial'** in lower left part of the Executive window, make sure **"All files in current directory"** be selected under **"File View"** in the middle part of the Executive window. (Refer to **"SACS Executive"** Figure)



Register model file **SACINP.DAT** to major task **Static Analysis** by right click the file and choose popup command **"Add to Project"**, choose **"Static Analysis"** under **Major Task** pull down window. Click OK to register. (Refer to **"Add File to Project Registry"** Figure)



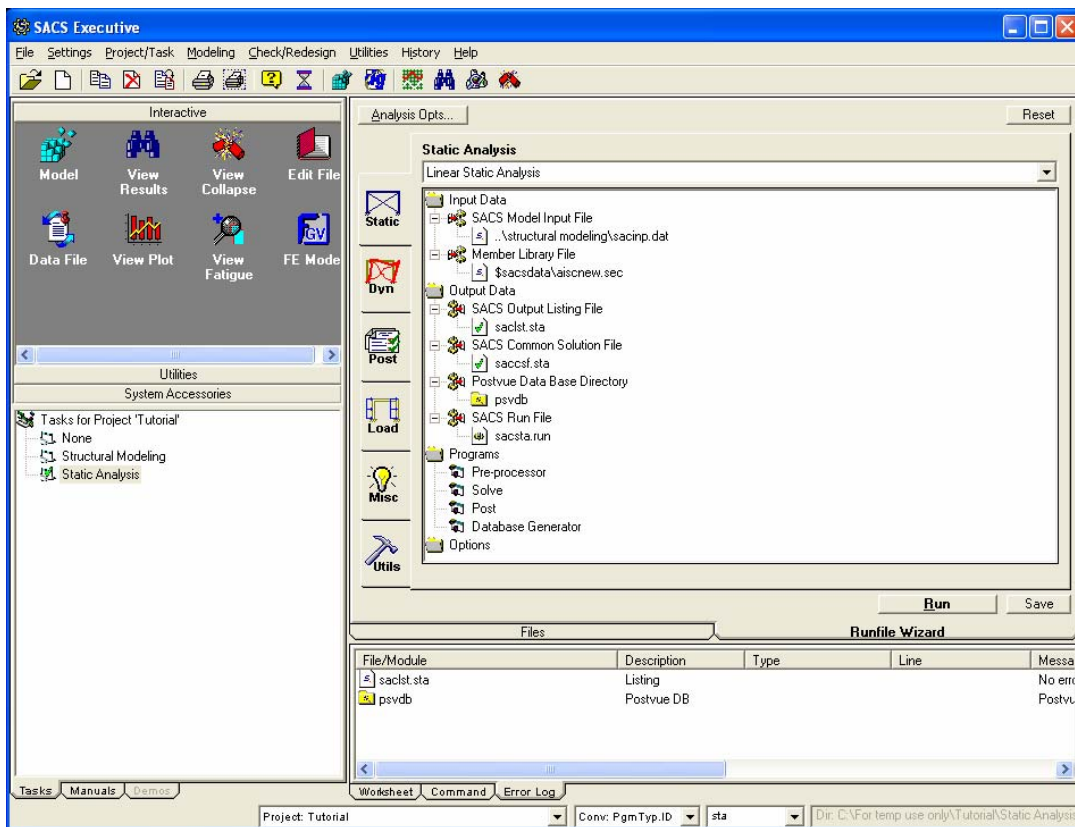
6. Create run file and perform analysis

Back to Major Task “**Static Analysis**” and make sure “**Files available to task ‘Static Analysis’**” be selected under “**File View**” in the middle part of the Executive window

Select “**Runfile Wizard**” and “**Linear Static Analysis**”

Click “**Start Wizard**” and choose “**Perform Element check**” and “**Postvue Database options**” then ok to complete.

Review each item and Click “**Run**” to execute the run. (Refer to “SACS Executive” Figure)



Please refer to the “**Run file Creating – Tutorial Problem.pdf**” file for more detail about this section.

7. Review results in both listing file and Postvue program.

This is the End of Tutorial Problem Part.

Appendix – SACS input file data deck for SACINP.DAT

```

1234567890123456789012345678901234567890123456789012345678901245678
Tutorial Frame
OPTIONS      EN      SDUC  2 1      C      PTPT      PTPT
AMOD
AMOD      3  1.33
SECT
SECT BOX10      BOX      10.000.250 10.000.250
GRUP
GRUP COL BOX10      29.0011.6036.00 1      1.001.00      N490.00
GRUP GRD W18X40      29.0011.6036.00 1      1.001.00      N490.00
MEMBER
MEMBER      1  2 COL
MEMBER      3  4 COL
MEMBER      2  4 GRD
JOINT
JOINT      1      0.      0.      0.      111111
JOINT      2      0.      0.      10.
JOINT      3      0.      20.      0.      111111
JOINT      4      0.      20.      10.
LOAD
LOADCN      1
LOAD Z      2  4      -1.0000      -1.0000      GLOB UNIF      DEAD
LOADCN      2
LOAD      2      16.0000      GLOB JOIN      WIND
LCOMB
LCOMB      3  1 1.000  2 1.000
END
***SPMB**      1  2      1  2  3  4      3  4  2  4      2  4
END
1234567890123456789012345678901234567890123456789012345678901245678

```