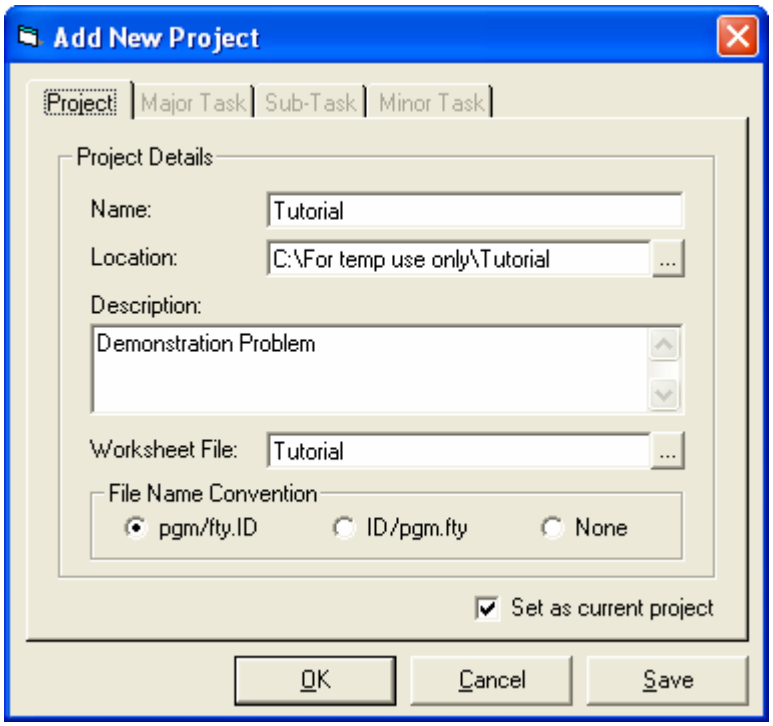
# Tutorial Problem

## 1. Model description:

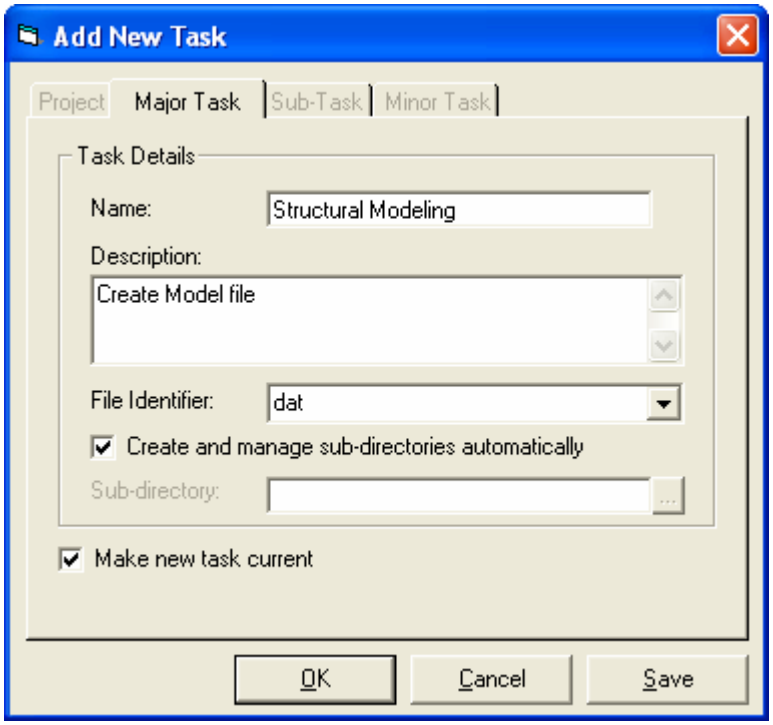
The frame is 10’ x 20’, the upright columns are 10” x 10” x ¼” box sections and the horizontal beam is an AISC W18x40 wide flange. Three load cases will be considered, Load case 1 is a uniform dead load of -1.0 kips/ft along the horizontal beam; Load case 2 is a 16 kips concentrated wind load; Load case 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)

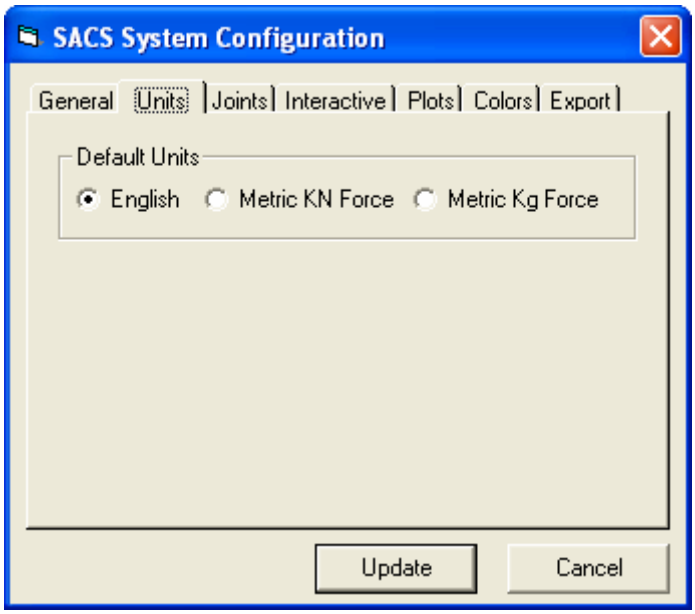



Under **Project/Task** menu select **Add major task**
Type in Name **Structural Modeling**
Type in description **Create Model File**
Type in File Identifier **dat**
**Then click OK (Refer to following** “Add New Task” figure)




### 3. Model generation using PRECEDE:

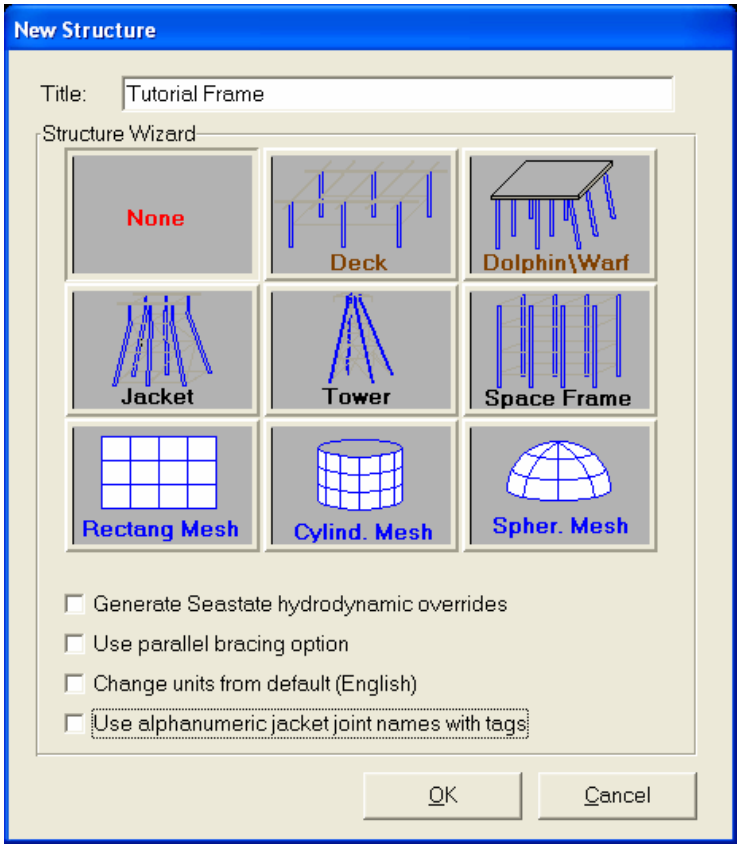
Make sure the unit is correctly set to English. Under **Settings > SACS System Configuration > Units**, select English unit if necessary. (Refer to "**SACS System Configuration**" figure)

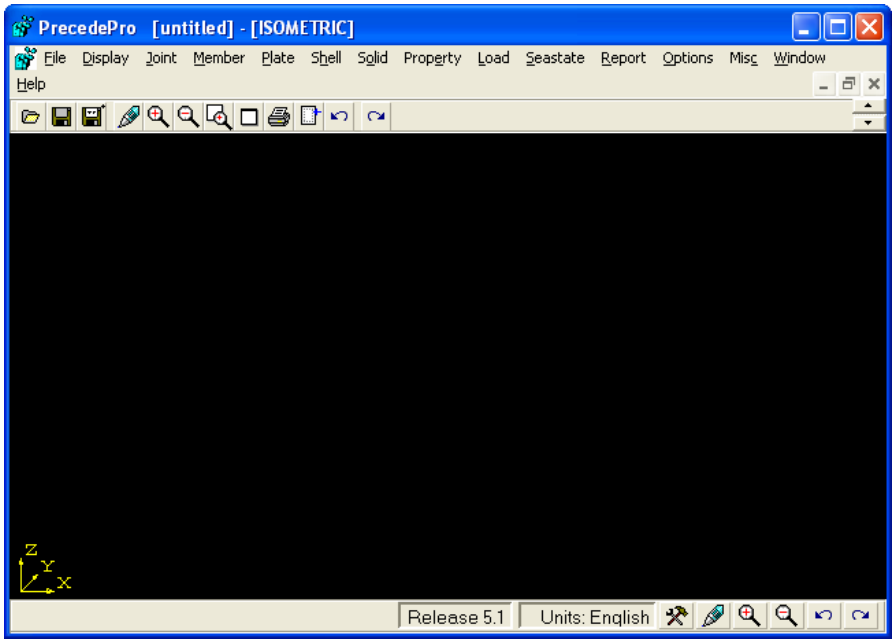


## a) Launch PRECEDE

Click **Modeling > Precede**
Select "**Created new model**" then **OK** (Refer to "**PRECEDE**" Figure)

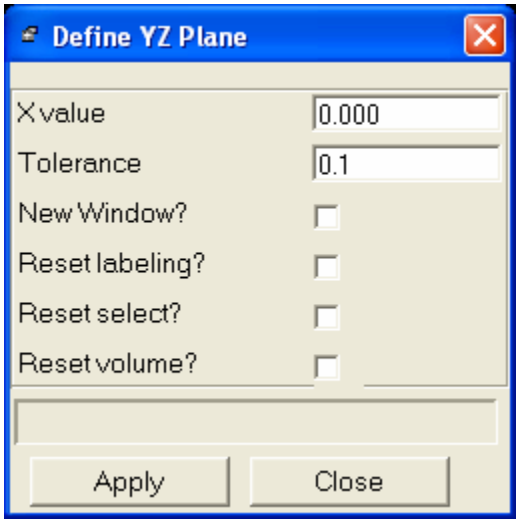

For new structure select "**None**";
Type in Title "**Tutorial Frame**"
Deselect "**Use alphanumeric jacket joint names with tags**"
Then click **OK** (Refer to "**New Structure**" Figure)
PRECEDE program will be launched (Refer to "**PrecedePro**" figure)






**b) Create joints**

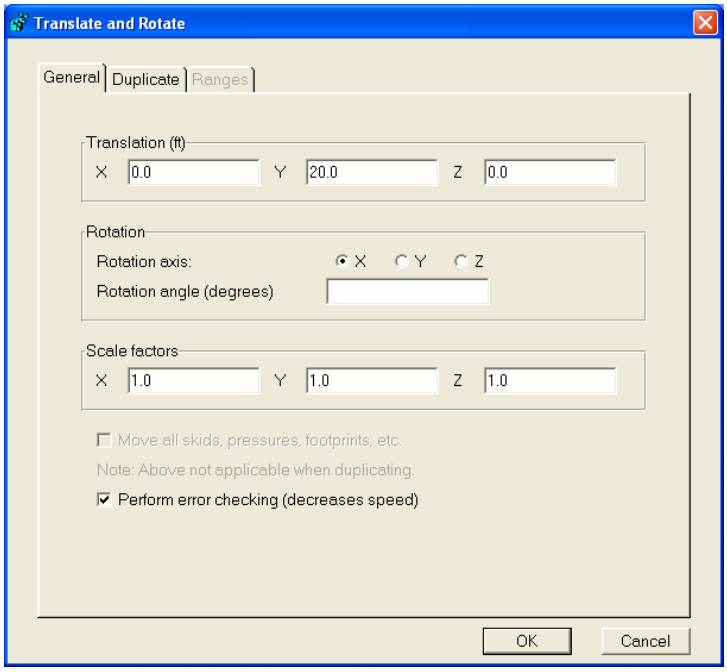
Add joint #1 with absolute coordinates of 0, 0, 0
Add joint #2 with absolute coordinates of 0, 0, 10.0
Use menu command **Display > Plane > YZ Plane** Change current 3d view to YZ plane, click any one of the two joints for X value coordinates. (Refer to "**Define YZ Plane**" Figure)

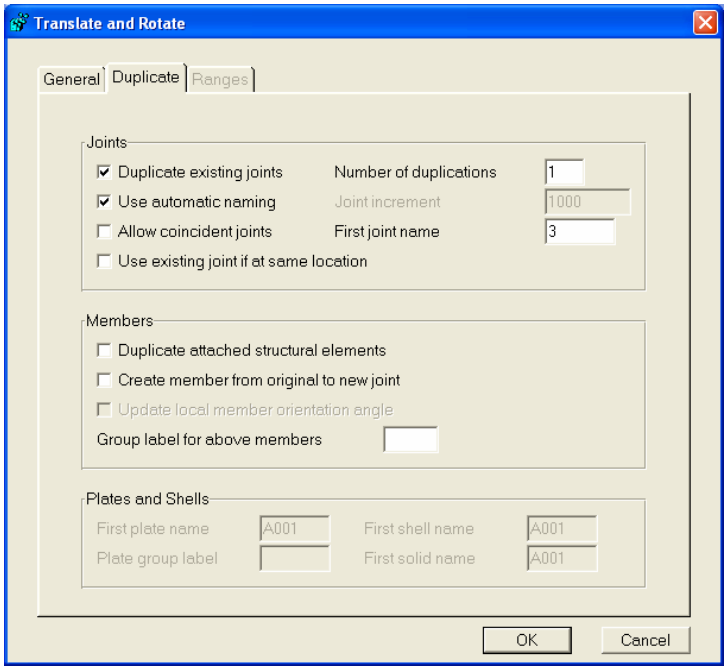



Use **Display > Zoom Box > Translate/Rotate > General** command
Select by drawing a window includes joint #1 and #2,
Type in Translation Y "**20.0**" (Refer to "**Translate and Rotate – Genera**l" Figure)
Select "**Duplicate**" then "**Duplicate existing joints**"

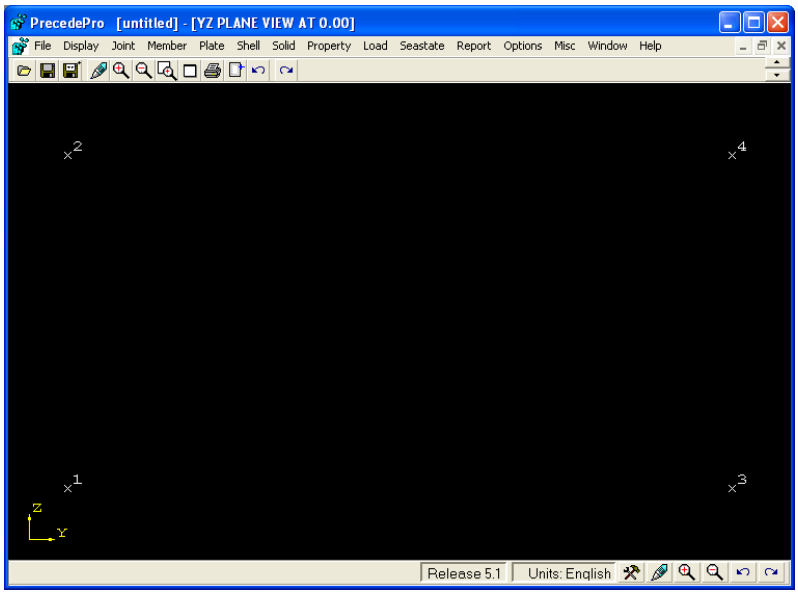
Type in Number of duplications “**1**” (Refer to “**Translate and Rotate – Duplicate**” Figure)

Click OK and created joints #3 and #4



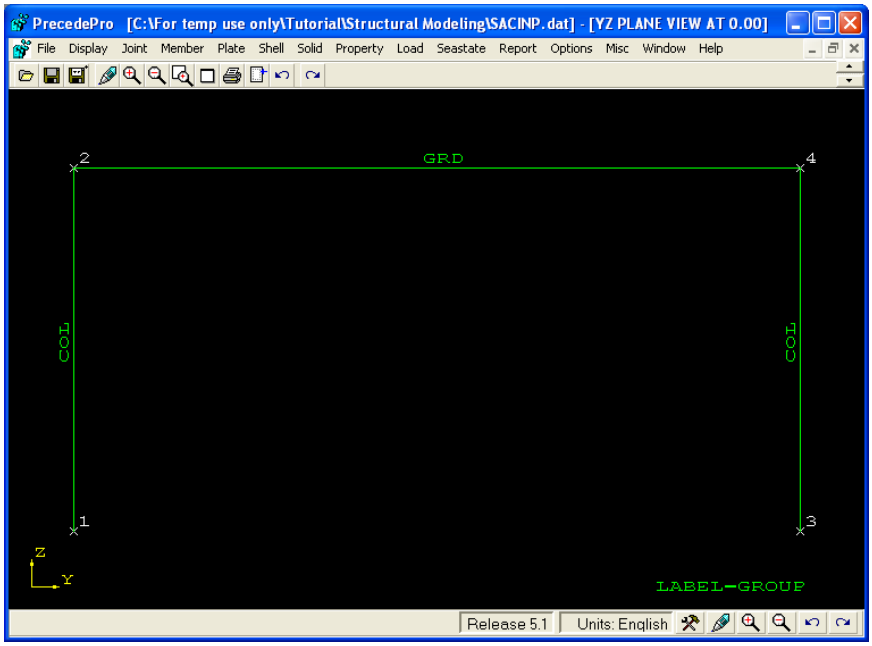

Use **Display > Labeling > Joint** to display joint names. We have created four corner joints of the frame. (Refer to “**PrecedePro [untitled]”** Figure)




**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**

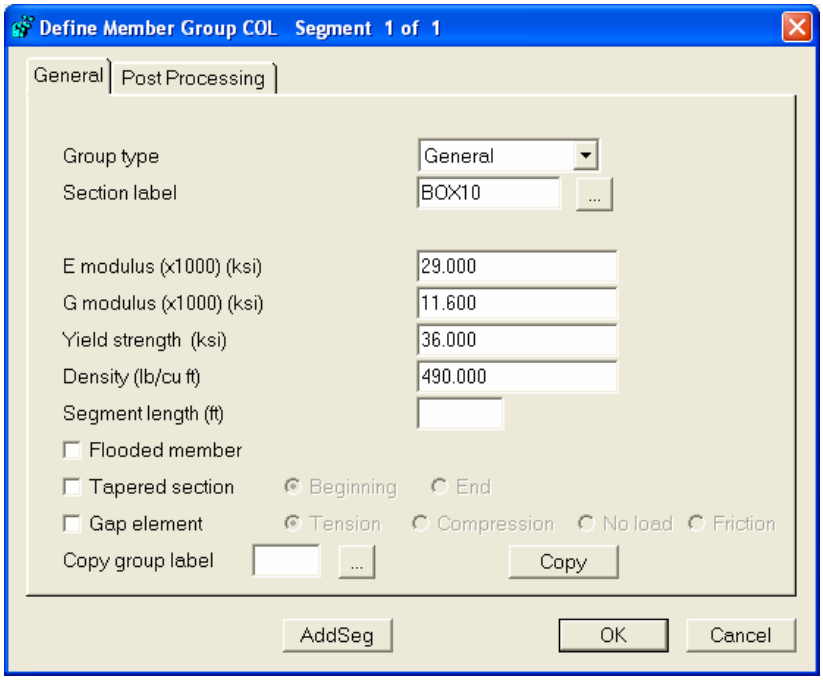
**d) Add frame members**

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 "**Precedepro [C:\...\Tutorial\Structural Modeling\SACINP.DAT**" figure)

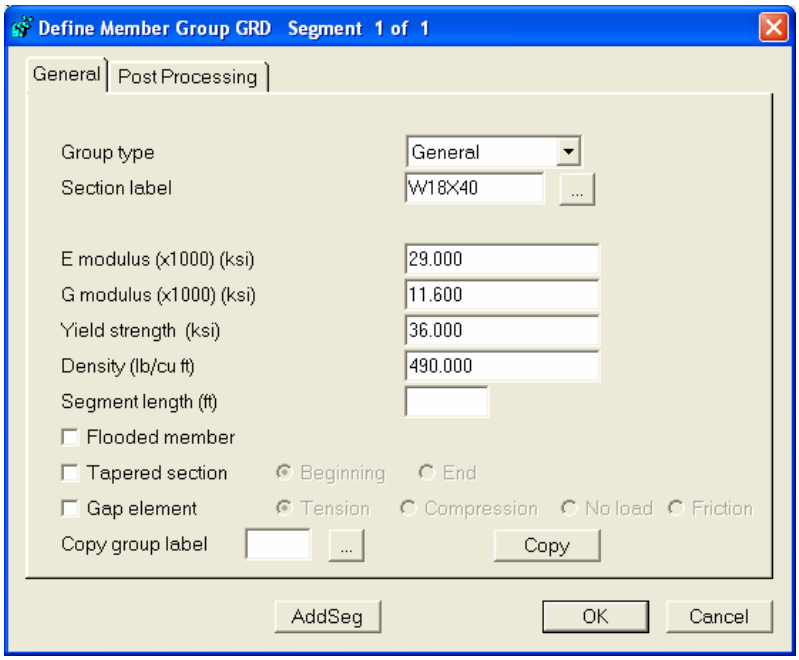


## e) Define member properties

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)



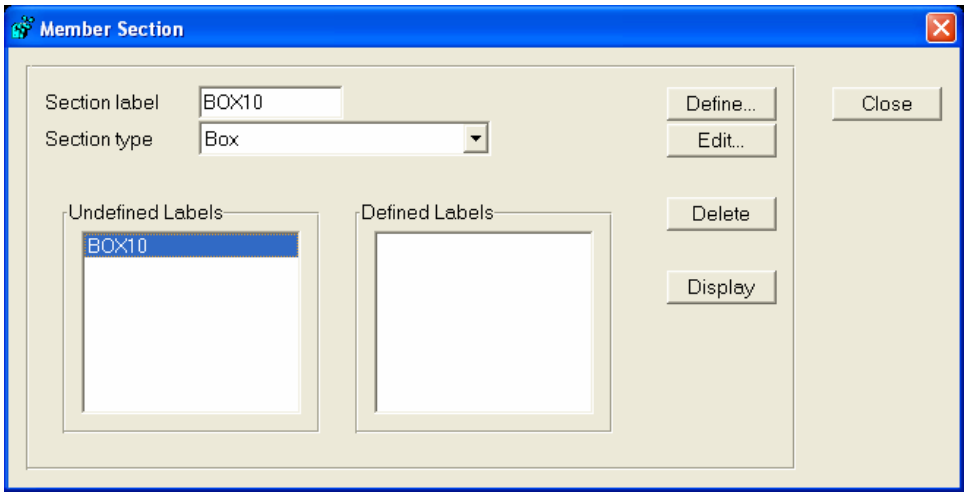

Choose "GRD" then "Define"
Select Group type as "General"
Browse in section label to find "W18X40" (Refer to "**Define Member Group GRD**" Figure)



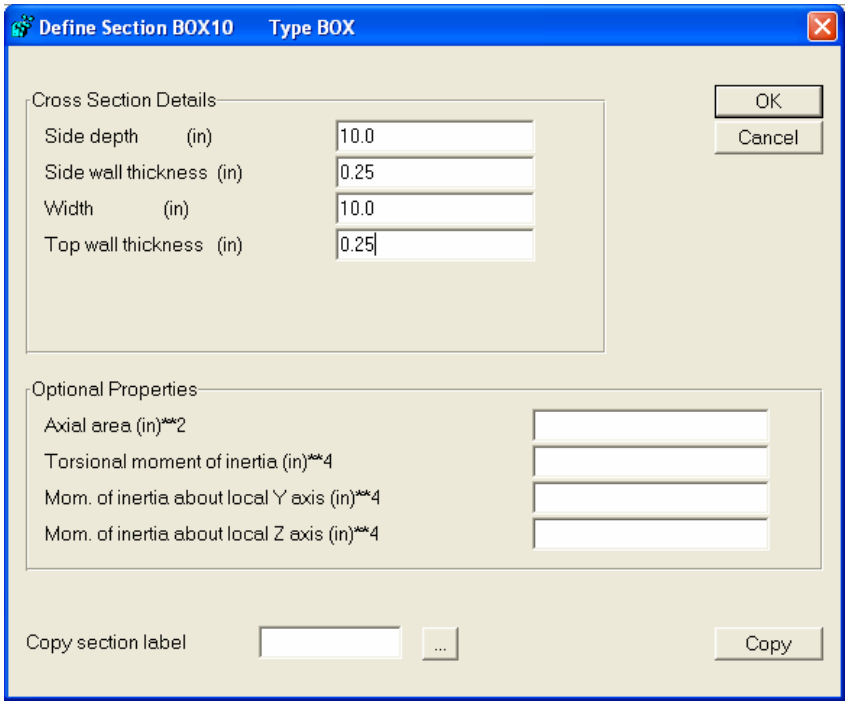
Select “**Close**” to quit member group definition window

## f) Define member sections

Select **Property > Member Section**
Choose “**Box10**”
Select Section Type as “**Box**”
Then “**Define**” (Refer to “**Member Section**” Figure)



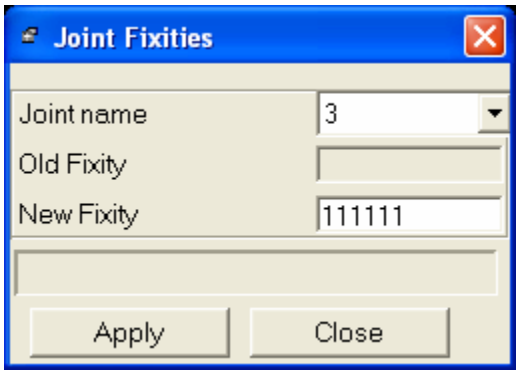

Type in Side depth “**10.0**”
Type in Side wall thickness “**0.25**”
Type in Width “**10.0**”
Type in Top wall Thickness “**0.25**” (Refer to “**Define Section BOX10**” Figure)



Then **OK** and "**Close**" to quit

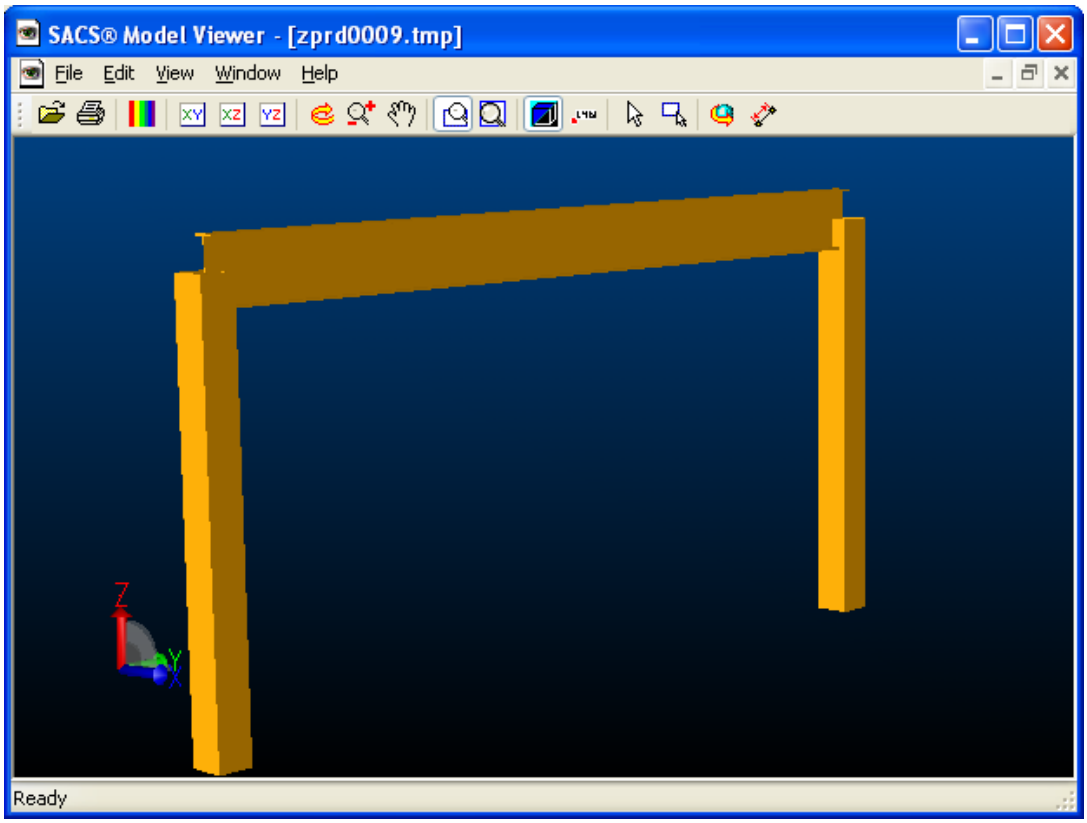
**g) Define joint fixities**

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)



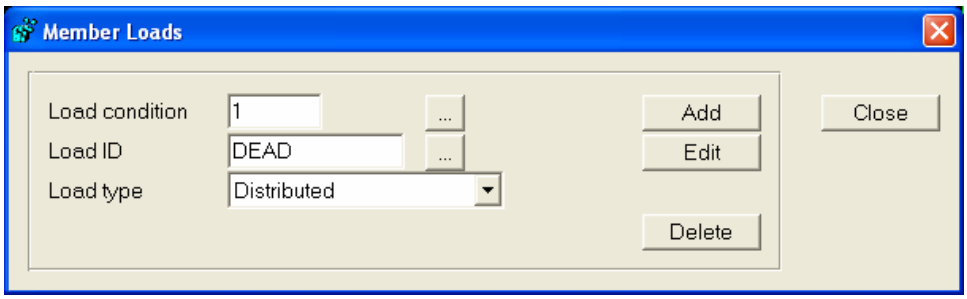

**h) Save and view 3D in Model viewer**

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

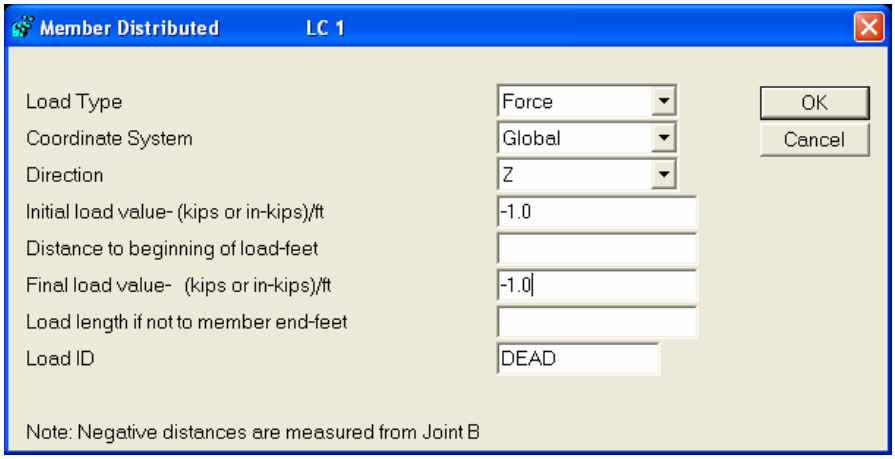


### i) Add load case #1

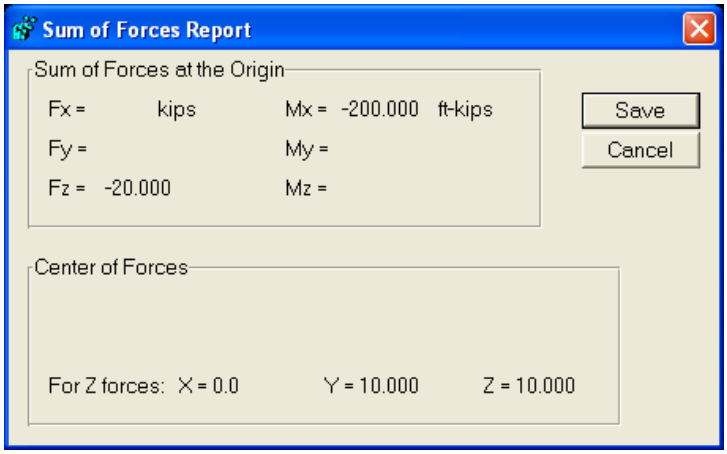
Select **Load > Members** then Choose member 2-4 and click "**Apply**"
Type in Load Condition "**1**"
Type in Load ID "**Dead**"
Select Load type as "**Distributed**" then Click "**Add**" (Refer to "**Member Loads**" Figure)




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

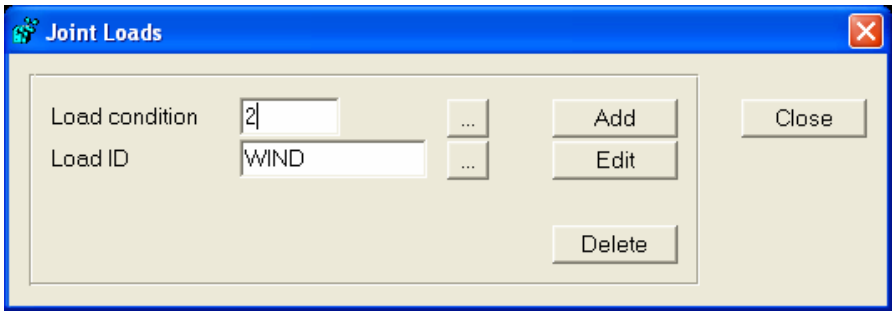



Review the force summary report and Click "**Save**", the load will display in the graphics window also. (Refer to "**Sum of Forces Report**" Figure)

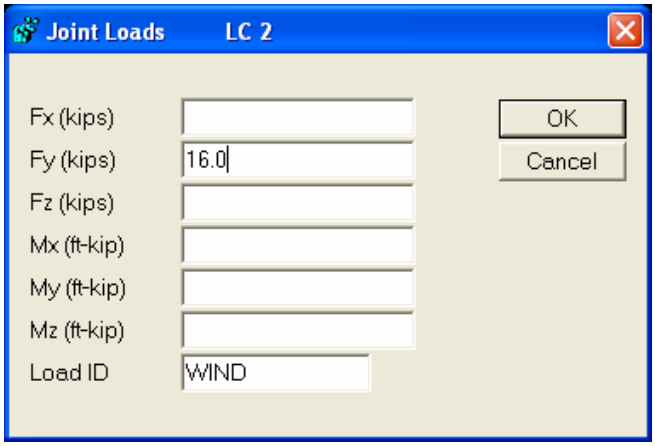


## j) Add load case #2

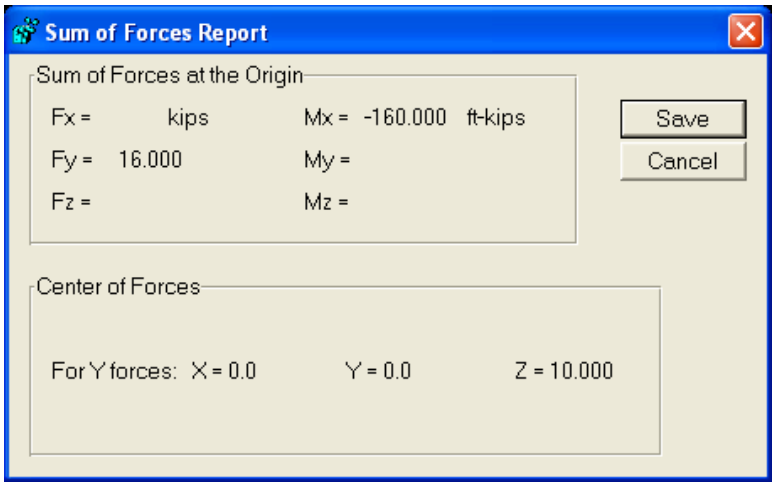
Select **Load > Joints** then Choose joint #2 and click "**Apply**"
Type in Load Condition "**2**"
Type in Load ID "**WIND**" then "**Add**" (Refer to "**Joint Loads"** Figure)




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

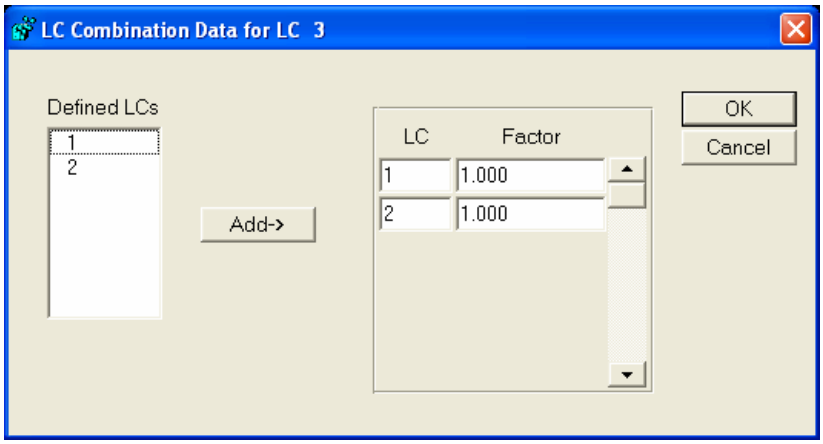



Review force summary report and "Save", note the force also shown in the graphics window. (Refer to "**Sum of Forces Report**" Figure)




## k) Add load case #3

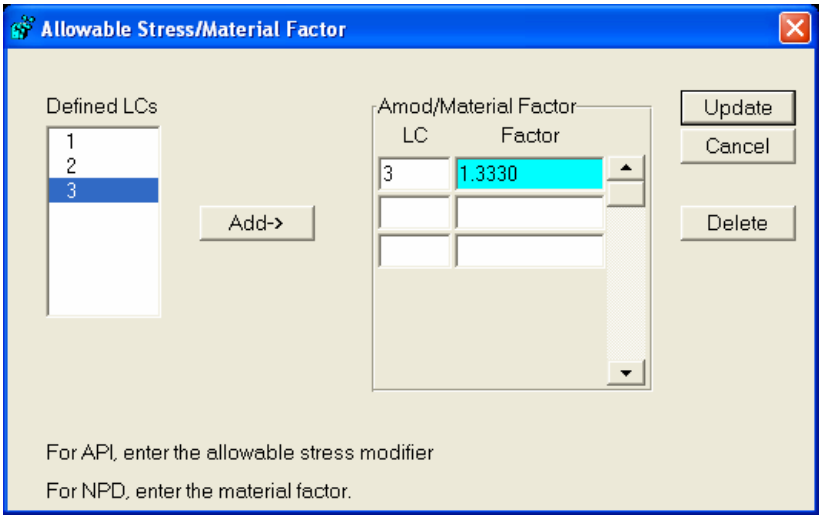
Select **Load > Combine Load Condition**s
Type in LC combination label "**3**" then "**Define**"
Choose Load case 1 and 2 then "**OK**" and "**Close**" to quit (Refer to "**LC Combination Data for LC 3**" Figure)




**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)

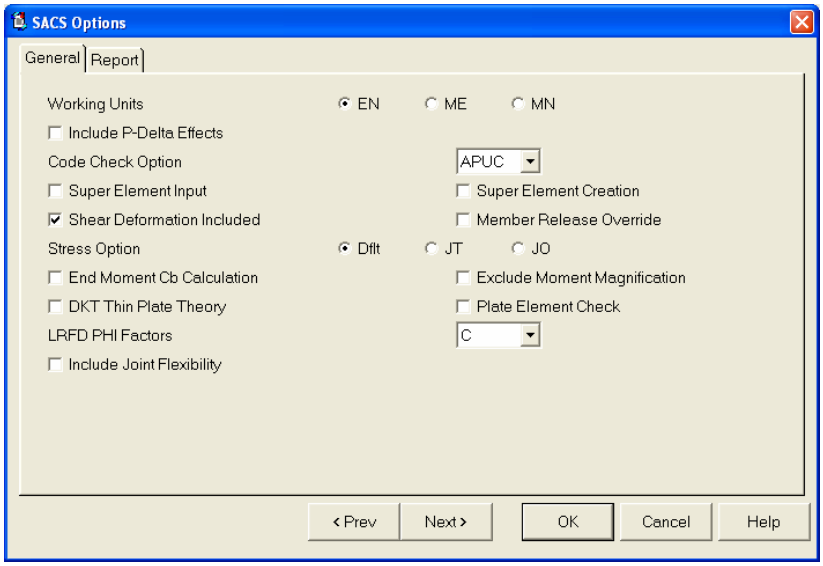



**Save**, **Exit** PRECEDE program.

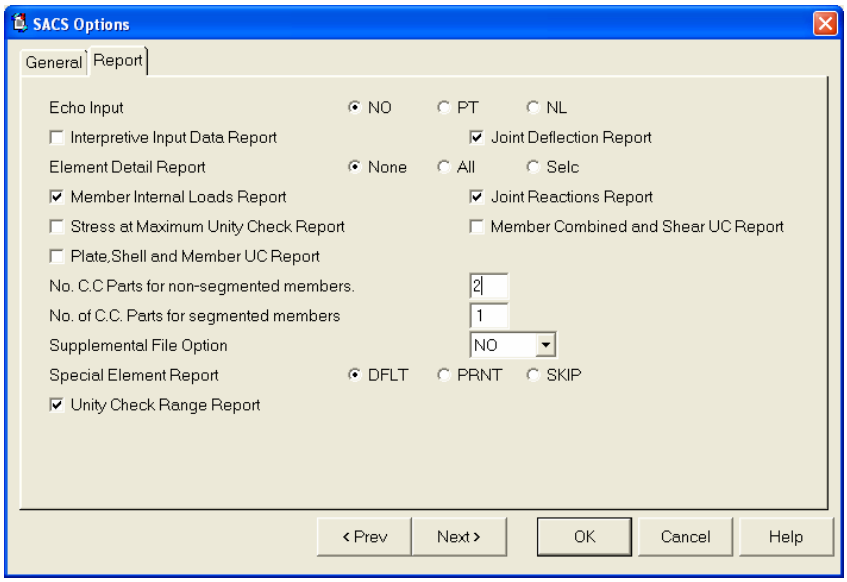
**4. Use Datagen to make modifications to model file SACINP.DAT:**

Choose **Modeling > DataGen,** select **Edit existing data fil**e and select **SACINP.DAT**.
Change options line: Double click the **OPTIONS** line, make necessary changes as

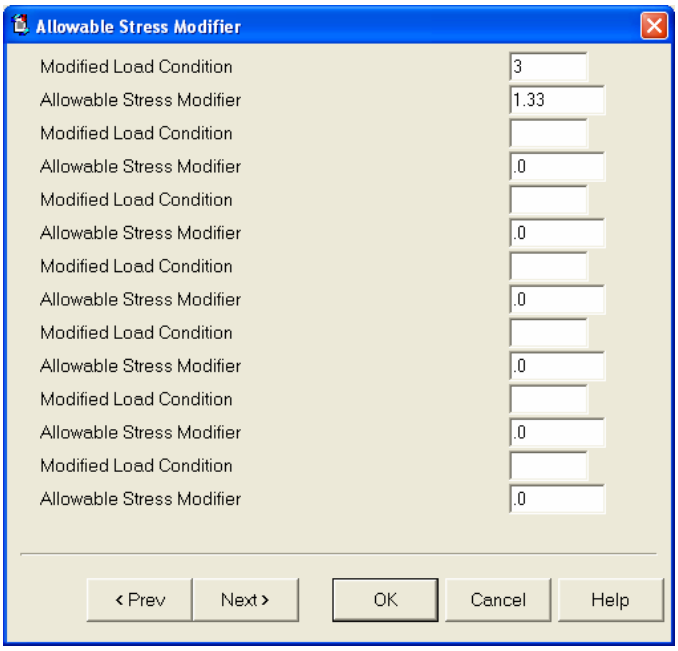
- General Window: select **APUC** for code check option – Tubular API 21st edition and others AISC 9th edition; (Refer to "**SACS Options – General**" Figure)




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



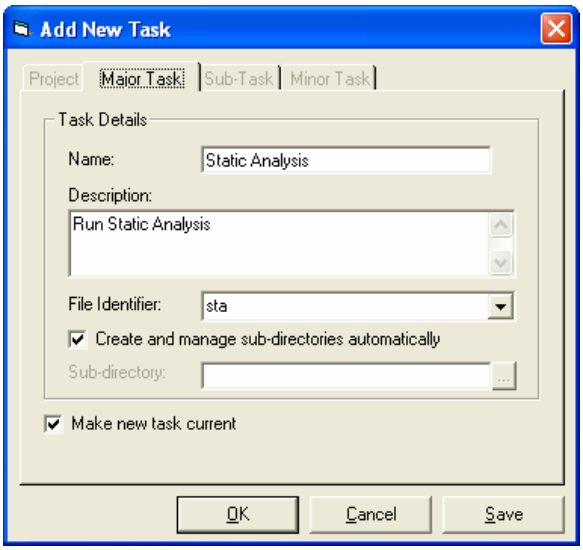
Double Click **AMOD** line and Change allowable stress modifier from **1.333 to 1.33** (Refer to "**Allowable Stress Modifier**" Figure)



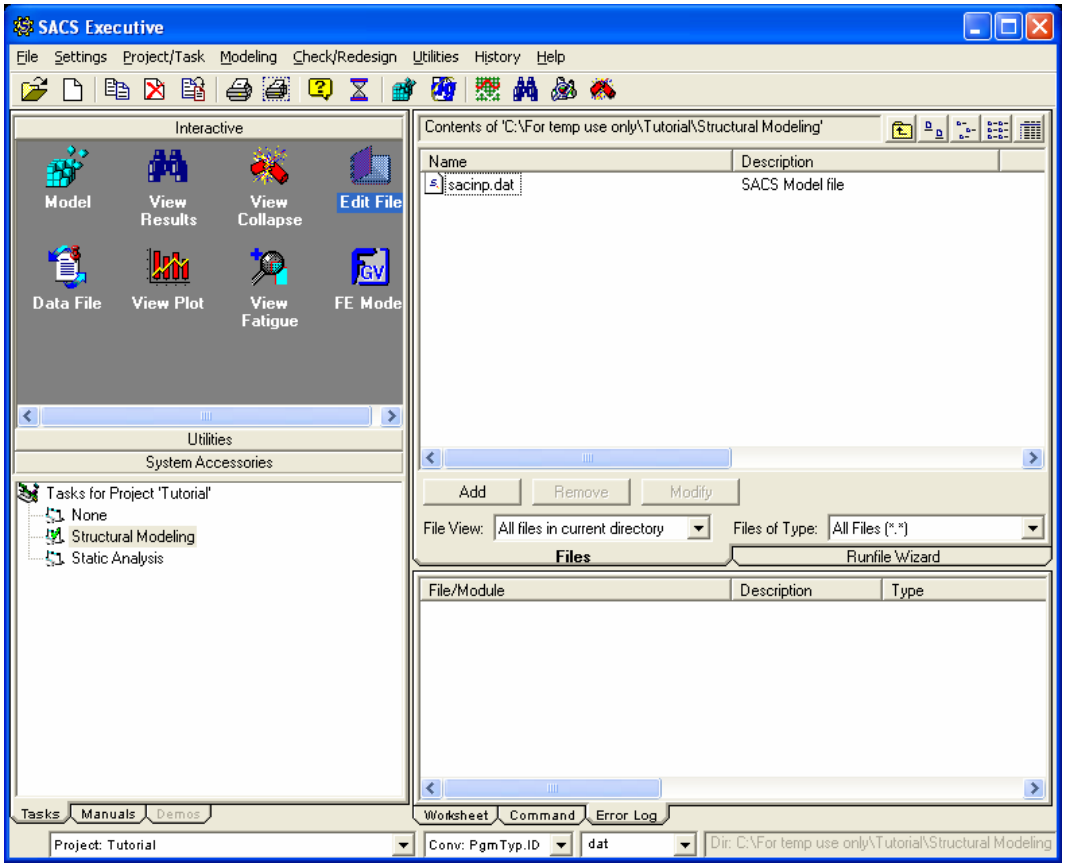

**Save** file and **exit** from DataGen program, the model file now is ready to run.

## 5. Create Major Task "Static Analysis":

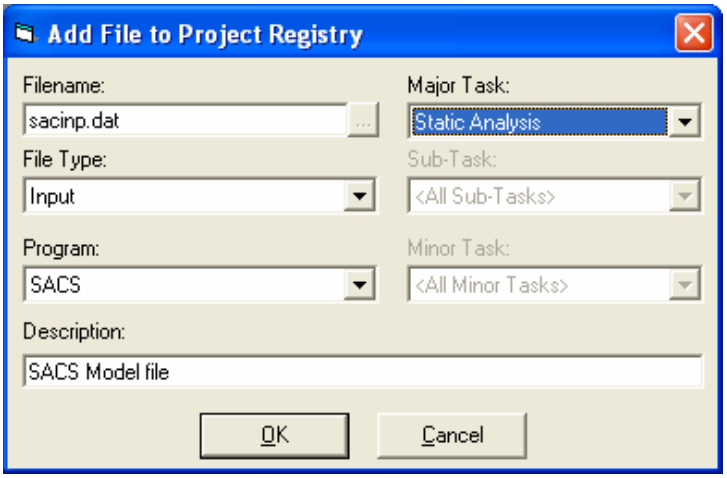
Use **Project/Task > Add major task** again
Type in Name "**Static Analysis**"
Type in description "**Run Static Analysis**"
Type in File Identifier "**sta**"
Then click OK (Refer to "**Add New Task**" Figure)



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)



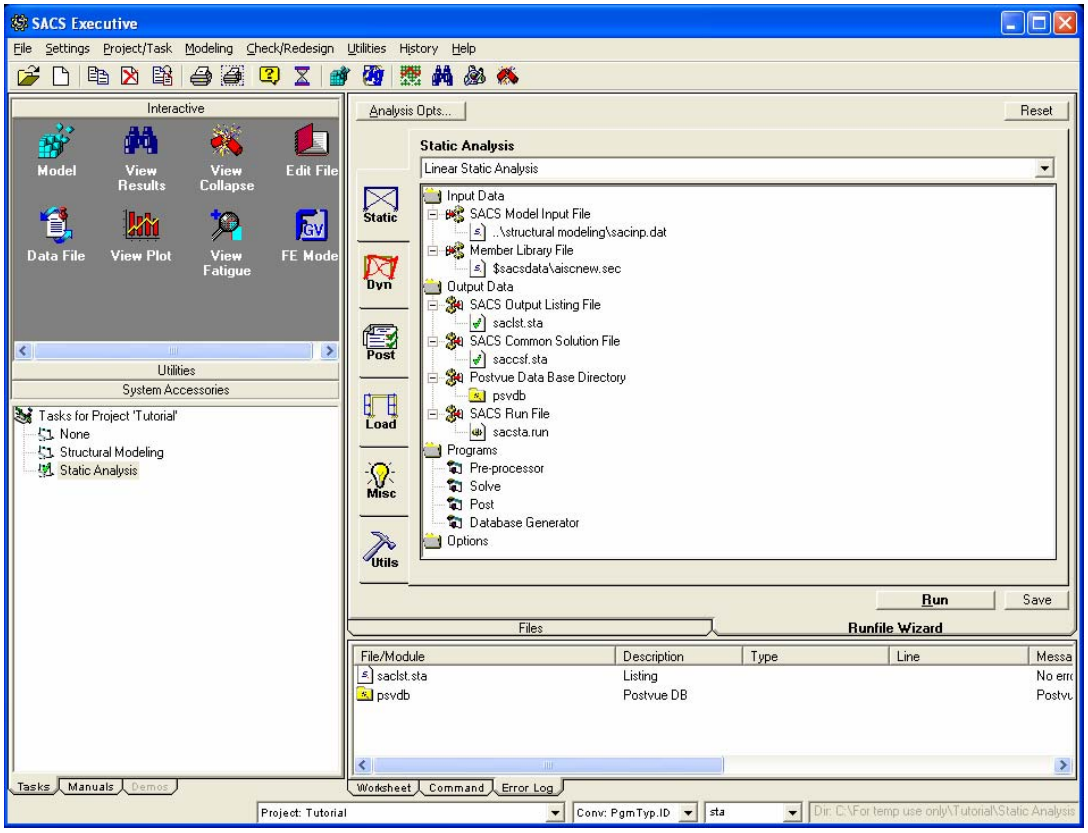

Register model file **SACINP.DAT** to major task **Static Analysis** by right click the file and choose popup command "**Add to Project**", choose "**Static Analysis**" under **Major Task** pull down window. Click OK to register. (Refer to "**Add File to Project Registry**" Figure)



## 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 Vie**w" 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

```
12345678901234567890123456789012345678901234567890123456789012345678901245678
 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
12345678901234567890123456789012345678901234567890123456789012345678901245678
```