New SACS 5.1 SP3 Features for StruCAD*3D Compatibility

The new release of SACS 5.1 Service Pack 3 includes new features compatible with the latest version of StruCAD*3D Version 4.3 Service Pack 1. These features include weight definitions, master/slave joint couplings and moving loads. These StruCAD*3D compatible features have been greatly enhanced to be more straightforward in understanding, more convenient in defining and more flexible in functioning.

1. Weights

1.1. Introduction

Various weights now may be directly defined in SACS via weight groups. These weight groups then may be used for further inertia load generation via the **INCWGT** “Weight Selection Data” line or passed to DYNPAC program for dynamic analysis via the **DYNMAS** “Dynamic Mass selection Data” line.

Weight groups may be defined as:

a. Basic Weight Lines

   By directly using **WGTJT** (Joint Weight Line), **WGTMEM** (Member Distributed or Concentrated Weight Data Line), **WGTNS** and **WGTNS2** (Non-Structural Weight Data Lines for multi joints), **WGTFP** and **WGTFP2** (Footprint Weight Data Lines) and **SURFWT** (Surface Weight Data Lines);

b. Weight Combinations

   By combining various pre-defined weight groups via the **WTCMB** “Weight Combination Input” line.

Weight density has been implemented throughout the full spectra of weight definitions. This powerful tool allows the user to account for hydrodynamic properties for various non-structural items in an extremely simple way.

Through the definitions of radius of gyration of corresponding weights where appropriate, another concept incorporated into the SACS weight definitions is the mass moment of inertia. This is very important for rigid body motions such as those encountered in transportation analysis and mass inertia effects in dynamic analysis.

The user can convert these weights into loads using **INCWGT** and/or **INCRAO** lines, accompanied by **ACCEL**, **MOTION** and **WAVE** lines. For example to determine the static loads from weight conditions, the user simply needs to specify the value of 1.0 (G’s) for the translational acceleration in the Z direction.

In Version 5.1 Service Pack 3 of SACS, all masses are entered in weight units rather than the mass units for the convenience of the user. Hence, reference is made to “Weights” rather than “Masses” throughout this document. All weight lines used in an analysis are grouped before the LOAD header input line.

1.2. Weights – User Defined Weights

User Defined Weights are completely defined by the user and each weight is applied to designated joint(s) or member.

- Joint Weight for a single joint is designated using the **WGTJT** line titled “Non-Structural Joint Weight Data”. The user has the option to input the weight density, global direction factors and even the radius of gyration for the mass moment of inertia;

- Member Weights can either be specified as “Concentrated Member Weight Data” or “Distributed Member Weight Data” using the same **WGTMEM** line. The user has the option to input weight density and direction factors globally or locally.

- Non-Structural Weight (Space Weight), using **WGTNS** “Non-Structural Weight Data” and/or **WGTNS2** “Additional Non-Structural Weight Data” line(s), the user has the option to define a weight in space which will be distributed automatically to up to eight specified joints. Except for options such as weight density, direction factors and radius of gyration, the user can make the non-structural weight refer to a feature referred to as the **ELEV** ID “Elevation Data” line. By using elevation IDs, the user can easily maintain the proper elevation.

1.3. Weights – Program Generated Weights

Program Generated Weights are created from commands that are more complex. These program generated weights are automatically assigned to the appropriate member(s) and joint(s) based on the information input by user.

- Footprint Weight (Equipment Weight) are equipment weights that are input using the **WGTFP** “Footprint Weight Data” and/or **WGTFP2** “Additional Footprint Weight Data” line(s). The user inputs the following information: 1. the weight, 2. the global coordinates of the center of the footprint, 3. the center of gravity of equipment either globally or with respect to the center of the footprint, 4. the number of skid beams (if any) in both the longitudinal and transverse directions, 5. the length and width of the footprint, and 6. the angle the longitudinal axis of the footprint makes with respect to the global X-axis. There are also functional options such as direction factors, radius of gyration, weight density and **ELEV** elevation identifiers used to change elevation.

With the use of footprint weight, the user has the option of eliminating secondary members from potential load transfer by specifying their group label on the **EXCGRP** "Exclude Group Data Line."

The footprint weight will work either with or without specified skid beams. With specified skid beams, the weight or load is transferred to the load bearing structural members as concentrated weights or loads. When the footprint weight is specified without skid beams, the weight or load will be transferred as member distributed weights or loads.

- Surface Weight, **SURFWT** "Surface Weight Data" allows the description of non-structural weights defined by weight pressure on specified surface areas. Two additional definition lines, **SURFID** "Surface ID" line and **SURFDR** "Surface Definition Data" line are necessary to complete the specification of the surface weight.

  The Surface ID line specifies a surface in 3D space, together with the surface boundary defined on the Surface Definition Data line. The program will make use of this information and automatically determine members to be loaded within the border of this region.

- Weight Combinations using the **WTCMB** "Weight Combination Input" line permits the user to combine factored weight groups. The user may combine both basic weight groups and other previously specified weight combinations as needed. The combined weight groups may then be referenced by **INCWGT** weight selection in specific load cases, or by **DYNMAS** dynamic mass selection to introduce these weights in dynamic analysis.

2. Space Force and Space Moment

The new **SFRC** (space forces) and **SMOM** (space moments) lines are additional features of the existing load definition family. These options allow the user to apply loads at a point in space, eliminating the need for having structural or non-structural members to define and distribute these loads. These forces/moments are transferred to the structure at as many as eight (8) defined joints.

3. Master/Slave Joint couplings

Master/Slave joint couplings define the Master/Slave joint translation and/or rotation couplings via the **MASTER** definition line. This feature allows the user to specify different joints in the structure that share degrees of freedom.

4. Moving Loads

Using the **MOVLOD** “Moving Load Case Selection Data”, **MOVGRP** “Moving Load Group Selection Data” and the **MOVSTP** “Moving Load Step” lines, the user has the capability to perform a Moving Load analysis within a load case. Using the information defined, SACS will generate a serious of load cases simulating the effect of a designated set of loads moving over the structure.

The MOVLOD line must be the first line in the group of lines defining a moving load case. The included load condition(s), which must contain only joint loads, must be defined previously.

The MOVGRP line allows the user to designate the member groups to which the moving loads are applied.

The MOVSTP specification allows the user to control precisely how the moving load group is “moved.” The moving load group may be moved in any direction and in variable or uniform intervals. The user has the option of defining the load movement direction either by specifying two different joints or the direction vector in the global coordinate system.

A moving load group includes all joint loads from all load cases included on the MOVLOD line. The initial position of the group is the location of the joint loads specified. The coordinate of the first joint load in the first load case specified in a moving load group is the reference point. All moving load groups are moved simultaneously from their initial position and will be moved along their own paths. Thus, it is important that the number of steps is the same in all groups. If different step values are specified, the minimum number of steps will be used for all groups.

LCSEL, LCFAC, AMOD or LCOMB lines may refer to the moving load case as a single load case, despite the fact that multiple load cases will be created from this one load input set.

5. Program Generated Inertia Loads

The SEASTATE program has been substantially enhanced to allow the user to generate inertia loads without running the TOW program.

The new SACS system allows the user to perform structural analysis for motion-induced inertia loads according to several methods. These include the **ACCEL** line and the **MOTION** line for basic load generation due to inertial motion, and the **WAVE** line accompanied with the **INCRAO** line for inertial load generation due to wave responses. Depending upon requirements, the user can select a simplified or a more rigorous approach for the load calculations.

5.1. Vessel Motion RAO (Response Amplitude Operator) Definitions

Using **RAO HEAD** and **RAO** specifications, the user now has the ability to input RAO's according to wave approach directions and vessel speeds. Furthermore, the user can input RAO's with their corresponding phase angles using displacement, velocity, acceleration or G input. This data refers to specific load cases using the **INCRAO** "RAO Include Data" line to generate motion response.

5.2. Inertia Load Generations – ACCEL line

The **ACCEL** "Acceleration Input" line enables the specification of translational accelerations, rotational accelerations and rotational velocities for the calculation of inertia loads within a load case. The weights to be included in the inertia load calculations are specified on the **INCWGT** line. Each **ACCEL** line refers to a center ID, which is defined by a **CENTER** "Roll Center Location" line. The user has the option of including or excluding structural weight in the inertia load calculation.

5.3. Inertia Load Generations – MOTION line

The **MOTION** "Motion Data" line enables the specification of motion characteristics such as roll angle and period, pitch angle and period, yaw angle and period and corresponding translational accelerations for the calculation of inertia loads within a load case. The weights to be included in the inertia load calculations once again are specified on the **INCWGT** line. Each MOTION line refers to a center ID defined by a **CENTER** line. The user has the option of including or excluding structural weight in the inertia load calculation. For MOTION inertia load generation, the user can also fully include weight gravity effects or include the effects of lateral forces only. This option allows the user to include DEAD loading in motion loading without having to remove the extra dead weight.

5.4. Inertia Load and Wave load Generations – WAVE line

In specific load cases containing **WAVE** definitions, an **INCRAO** line will include the inertia load calculation procedure for wave responses. The weights to be included in the inertia load calculations are specified on the **INCWGT** line.

Three different interpolation procedures are utilized:

- Vessel Speed
  If the vessel speed defined in the **INCRAO** line does not correspond to a given RAO vessel speed, then interpolated or extrapolated RAO values are used;

- Wave Approaching Direction
  If the wave approaching direction specified on the **WAVE** line does not correspond to a given RAO wave approaching direction, then interpolated RAO values are used; and

  Wave Period

  If the wave period specified on the **WAVE** line does not correspond to a given RAO wave period, then interpolated RAO values are used.

For structures partially or wholly submerged in water or affected by the passing of a wave crest, wave loads and inertia loads will be calculated simultaneously.

StruCAD*3D to SACS Model File Conversion Exceptions

The StruCAD*3D Version 4.3 Service Pack 1 file conversion facility available in SACS 5.1 Service Pack 3 can convert the following StruCAD*3D model files into SACS model files:

1) Noah data file for linear static analysis
2) Noah data files with SSI data.

Exceptions: For Noah data file the following commonly used inputs are not converted into the equivalent SACS model:

1) Non-structural elements including NONSTR, NSGRP, JDIST, MDIST, SPGRP etc.
2) Encased material specifications using ENCASE input.
3) Anode zone elevation definitions using ANODE input.
4) User defined wind profiles using WNDPRO input.
5) All dynamic related options including loads, spectral wave specifications and dynamic input for pile linearization.
6) Progressive constructions analysis option.

The StruCAD*3D program also has several supplemental analysis files that are not presently converted into SACS model files and these include:

1) Post data files.
2) Pile Data files.
3) Float/Launch data files.
4) Fatigue data files.
5) Update data files.
6) Joint Can data files.
7) Motion data files.

Note: All of the above **Exceptions** can be manually converted into equivalent SACS model files.