

ISYMOST Release Notes

ISYMOST Software Distribution

These Release Notes refer to the Software Distribution of ISYMOST.

ISYMOST is an interactive graphic software program, pre-processor and post-processor designed for calculation codes using finite elements

The software is made available under license to Customers who have accepted the license terms and have paid the corresponding fees. The Software Distribution includes executable files and libraries, configuration files and documentation.

Copyright Information

The ISYMOST software is protected by Copyright laws worldwide and all or any part of this Software Distribution belong to ISYMOST, unless explicitly stated.

© Copyright PRINCIPIA 1992-2011

All Rights reserved. No part of the Software Distribution of ISYMOST may be disassembled, reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without the permission of the copyright owners.

Please refer to your license agreement for details.

System Requirements

The present version of ISYMOST can be run on

- Intel x86 based PCs and servers under any of Windows XP Pro SP3, Windows 2003 server, 2008 server, Seven Pro.

Versioning

The ISYMOST software is developed and maintained continuously. As a result, successive releases of the software are made available to licensed customers. Identification of the software release is a numeric string of the form v r m, where

- v is the version number,
- r the release number,
- m the modification level,

For example the release 0 of version 5, modification level 1 could be identified like v501.

A development index (*_di*) or a patch index (*_pi*) could be supplemented if needed.

The current version of the software is the topmost reference in the Summary of Changes section.

Database compatibility

Databases remain upward compatible until version 4, release 4.

Summary of Changes in v505

Path to solver executable files

- Path to solver executable files (NSO, DIODORE, ABAQUS®) and ISYRET are defined in OCEANOS_USER.INI file, and no more in the ISY_SOLV.BAT file. These paths are read when CALCULATE is run from Analyses Menu.

3D VIEWER Improvement

- Parameter ACCEL in OCEANOS_USER.INI file to improve the 3D behaviour when the video card is integrated on the motherboard (ACCEL=2). Default is ACCEL=1 for higher-performance graphics card.
- Animation on 3D viewer is enabled.
- Automatic centering of the view when selection is changed from Objects tree.
- Improvement of node selection.

Other Improvements or Corrections

- Improvement of Buckling Lengths definition.
- Correct menu for tangent axes.
- Correct value of angle for Edges command.
- New symbols for unilateral supports.
- Computation of Theta angles when only 1 direction is given.
- Accounting for eccentricities and end joint sizes when computing beam weight.
- Tubular intersections with only 1 chord member.
- Tubular intersections with tubes described through tables.
- Suppress « Category » when creating Material or Physical properties.
- Add a flag to enable or disable User's tables.

Interfaces

- Save working directory for ISYRET.
- Management of nodes labels (A4) for models retrieved from SACS®.
- SACS® sections: PGB, PGT...
- Improvement of ISYRET for ANSYS® (up to release 12.1).
- Correct error with member eccentricities for ABAQUS®.
- Management of Elsets for ABAQUS® load cases.
- Reading of ABAQUS® results for Frame3D and Pipe31 beams.

New Features

- Recursive calculation of combinations using other combinations as components
- Computation of centre of loads.
- Physical property for Cones.
- Creation of Pipes and Elbows.
- Pressure in pipes (internal/external, hydrostatic, end status: close or open).
- Results Dialog Box.
- RHS Intersections.

Results Dialog Box

- A new dialog box has been created for the handling of results. It enables to display results from static, modal and non-linear analyses. This dialog box is open or closed by typing "F4", "R" or selecting "Results" in Main menu. It does not work for sorting or coupling features.
- The Results control panel is still available, with the same functionality as previous releases. It is open or closed by typing "Ctrl+R" or selecting "Results" in Main menu.

Summary of Changes in v504

UNDO /REDO

- Modifications of nodes, elements, objects, loadings and analyses are recorded and can be undone or redone.
- 10 successive modifications are stored and can be undone.

Toolbars

- 2 toolbars have been created in the Graphical User Interface.
- The top toolbar is dedicated to the display parameters. It replaces the Display menu and contains other facilities as undo/redo buttons, selection's filter, 3d-viewer functions...
- The bottom toolbar is used to handle the coordinate systems.

Groups & Objects Tree

- A tree representation is used for groups and objects. It replaces the Groups menu.
- "F3" is used to give focus to the Groups window.

New Results Dialog Box

- A tree new dialog box has been created for the handing of results. It enables to display results from static, modal and non-linear analyses. It does not work for sorting or coupling features. This dialog box is open or closed by typing "R". It is distributed as a beta feature in this release.
- Results control panel is still available, with the same functionality as v5.03. It is open or closed by typing "F4" or selecting "Results" in Main menu.

Units and Viewbox Dialog Boxes

- New dialog boxes are available for Units and Viewbox.

Improvement in 3D Viewer

- When printing from the 2D-viewer, the picture is sent to the printer and a bitmap file is created.
- New functionalities have been added to control the point of view:
 1. Change centre of rotation, by selecting a node (use "J" to cancel),
 2. Fit all and Fit view,
 3. Adjust to the nearest predefined point of view (Snapping, "S")
 4. Use "CTRL+SHIFT" and right button to rotate about screen axes, and "CTRL" to rotate about screen axes.

Parameters

- Effective length factors (KY, KZ), unbraced length of compression flanges (UNLCFY, UNLCFZ) and distance between web stiffeners (WEBST) are created in Isymost.
- KY and KZ may be defined in physical property objects or on elements. Other parameters are defined on elements only.
- The parameter PROFILE is set to RHS for beams with square or rectangular hollow sections.

G

- The value of G (gravity) is set to 9.80665 to be homogeneous with NSO.
- The previous value (9.81) may be used when translating data to NSO if the parameter G981 is set to Yes in isytrad.res file.

Tables

- Tables for JIS, UC and UB sections are available.

T & L stiffeners

- T and L sections can be declared as stiffeners when creating them. In this case, the correct I/V value is used and the eccentricities are automatically computed.

Details for sorted values

- T and L sections can be declared as stiffeners when creating them. In this case, the correct I/V value is used and the eccentricities are automatically computed.

List/Info

- Commands List and Information are available for alphanumeric identifiers and tangent axes of plate or shell elements.

Interfaces to other software

- PDMS files may be output (beta version).
- ABAQUS: combinations of loading cases are translated, and member releases (for rotational DOF only).
- SACS: several improvements. Management of nodes with number greater than 9999.

Commands renaming

- For clarification, some commands have been renamed:
 1. HELP becomes COMMANDS
 2. RECOVERING becomes OVERLAPPING
 3. OVERDRAW becomes SUPERIMPOSE
 4. MAP becomes LEGEND

Summary of Changes in v503

Default Objects

- Default objects may be defined for material and physical properties (beam, plate and spring). If the default objects are defined, elements are automatically affected in these objects when they are created, and are affected to "NUL." In all other material and physical properties groups.

Beam Modelling

- Check for crossed beams (Main Menu / Check / Check / Mesh): this function detects the pairs of beams that must be subdivided.
- Merge of beams (Main Menu / Modify / Merge): 2 aligned beams, having the same properties, can be merged into a single beam if the common node is not used by another element. Loads on the beams are modified.
- Subdivision of a beam by another beam: the intersection node between the 2 beams is created, and then the 2 beams are subdivided if the intersection is on the beam, or lengthened if the intersection node is outside the beam. In case of lengthening, a new beam is created, with the same characteristics as the lengthened one.
- Generation of member end joint sizes.
- Display of beam slenderness (Results Menu / Load Data / Internal Variable on Elements / Geometric Variable).
- Management of beam buckling lengths in true length instead of factor.

- Modification of beam parameters (releases, eccentricities and buckling lengths) by means of the mouse's right-button.

Display of Results

- Detailed output of code check results by means of mouse's right-button (refer to NSO "Create Isymost Report" command).
- Inversion of the coloured scale used for iso-values (Display Menu / Annotate / Iso-values). Useful to emphasis negative values (compression for example) or small values (lifetime for example) instead of large values.

Ergonomics

- Transparency of objects may be defined for 3D viewer (mouse's right-button on an object).
- The option "Elements in a plane" is added into the elements selection menu.
- **isymost.res** file may be used again to set the value of some graphical parameters, which are not set in the **Oceanos_User.ini** file. Please refer to Programmer's Manual for more details.
- In addition to the 5 binary save files automatically generated by Isymost (*jobname_1.ISB* to *jobname_5.ISB*), the user may created his own binary save files, such enabling him to retrieve a particular stat of the database (Edit / Binary Save / File name). These files are not deleted at the end of the Isymost session.

Software Interfaces

- DXF files may be output.

Miscellaneous

- Numbering of tubular intersection elements is now controlled by the user in the same way than structural elements.
- When empty objects are deleted, analyses are modified instead of being initialised.
- When material or physical properties are checked for an analysis, the elements with no defined properties are displayed and selected. The check command is on the line "Analyzed Structure" in the Analyses Menu.

Summary of Changes in v501

3D-Viewer

- Creation of a new 3D-Viewer, based on OpenGL library.
- All creation, generation or visualization operations may be performed using this viewer.
- Compatible with 2D-Viewer.

Coordinate Systems

- Definition of New Origin and/or New Axes.
- Use of Cartesian, Cylindrical or Spherical coordinates.

Ergonomics

- File names have no more length limitation (up to 256 characters).
- Groups, Objects, Loading Cases and Analyses identifiers are case sensitive.
- Review of SORT command, to facilitate the selection of loading cases.
- Management of BDR files for enabling the overwriting by a solver while displaying by ISYMOST.
- Add LIST TABULAR command for Physical Property Groups: writes the list of object in a tabular format suitable for input to a spreadsheet program.

- When calculating the resultant of forces, masses or pressures, the results may be output object by object.

File format

- ISYMOST file format has slightly changed, and files created with v5.01 cannot be read using v4.70 or older versions. Use "Save v4.70" for compatibility.

Initialization and Resources

- ***isymost.res*** file is not used anymore.
- A number of parameters are initialized in ***Oceanos_User.ini*** (see Programmer's Manuel for details).
- Groups of commands for analyses are stored in ***isytrad.res***.

Summary of Changes in v470

Data and Results Management

- Automatic generation of spring supports on a surface or a set of beams
- Calculate and change the resultant of a set of springs.
- Computation of edges with respect to the objects of a group
- Control of bounds when displaying values of elements
- Improvement of processing of harmonic results
- Addition of a section ".SCALNOE" in BDR file, enabling to display any result at nodes
- Display of magnitude for displacements or reaction forces
- Display of shear stress criterion for beams
- Display of maximum principal stress for finite elements

Structural Software Interfaces

- New ISYMOST/SACS and SACS/ISYMOST Interfaces

Ergonomics

- Contextual menu for Nodes, Elements or Objects
- Called by means of mouse's right-button
- Filter for selection in the main menu