Mechanical Engineering News

FOR THE POWER,

PROCESS AND

RELATED INDUSTRIES

COADE
ENGINEERING SOFTWARE

VOLUME 35   **DECEMBER 2004**

COADE, CAESAR II Celebrates 20 Years
(by: Tom Van Laan)

In 1984:

- a second superpower (the Soviet Union) was presided over by Yuri Andropov

- prominent in American popular culture were "The A Team" (television), Culture Club (music), and Haing Ngor (movies)

- computer technology raced along, with the introductions of IBM's PCJr and Apple's Lisa 2

- very few had yet heard the words Internet, SUV, hip hop, or reality TV

IN THIS ISSUE:

CAESAR II Version 5.00

COADE's Plant Design Strategy

Stress Analysis of Skirt Supports with Openings

B31.3 – 2004 and CAESAR II

What's New at COADE

Technology You Can Use

Program Specifications

1984 seems like a very different time than that which we experience today. But there were some similarities:

- despite record budget deficits, a sluggish economy, and American soldiers under fire in a Middle Eastern nation (Lebanon), George Bush (Sr.) was elected on a national ticket to a second consecutive term in office (as VP)

- COADE's CAESAR II reigned as the leading PC-based pipe stress analysis software

Yes, it is true – COADE and CAESAR II are now 20 years old, and this month – December 2004 – we celebrate the anniversary of the first sale of CAESAR II, a sale of two licenses made to SOHIO, whose pipe stress engineering department only owned one PC!

Since 1984, the world has changed considerably, and companies and products have come and gone (for example, SOHIO, the first CAESAR II customer, ceased to exist as an independent entity less than three years after that initial CAESAR II purchase).

But one thing has not changed – COADE has continued to grow and move forward, adapting to changing technology, changing engineering requirements and expectations, and the changing business climate and marketplace. We are proud to be where we are today, 20 years after that initial sale:

- serving more than 22,000 licensees in more than 6500 companies in 65 countries

- having experienced 19 years of increasing sales

- providing not just CAESAR II, but also CADWorx, PVElite/CodeCalc, and TANK (premier products in the world of AutoCAD-based plant design, pressure vessel analysis, and API 650/653 analysis and design, respectively)

- reaching the world through our 50-country distribution network

Thanks for this success are due to all of the employees who have contributed to COADE since the beginning – there are too many to name here. But more importantly, thanks are due to all of the customers and prospects over the years who have been willing to take the time to consider our offerings and give them a fair evaluation. Without the confidence that you all have shown in us, we would not be here to celebrate our 20th.

COADE Staff, ca. 1990

Selected COADE Staffers Today, Older and Wiser

New Features in PVElite Version 2005

(by: Scott Mayeux)

PVElite 2005 represents a significant product update over the previous version. One of the most notable changes in the new version is an updated user interface. The Element Input Data, Design Constraints, Title Page Data and Wind/Seismic have all been organized into a tabbed property view. The tab containing the Input data along with all of the other toolbars are dock able and now retain their locations in between sessions. Much of the input data now appears in a grid-based format. The up and down arrow keys along with the tab and enter key allow movement in between the data fields. The new input screen is shown below.

A few other features of this new version are:

- ASME Code 2004 Updates, including UHX and the Appendix 26 updates

- All new Material Database Editor for ASME, PD:5500 and EN-13445

- EN-13445 Material Database

- PD:5500 Yield Stress Database

- Updated Output Processor

- Enhanced Word Generation in the Output

- Metric Bolts have been added per TEMA, British Standards and South African Standards

- Several Structural Steel Databases have been added including Korean/JIS, British, German, South African and Australian. These databases will be available for leg and stiffening ring design.

- ASME Steel Stack Design per STS-2000 has been implemented

- External Corrosion Allowance is accounted for

- Nominal Thickness Input is available

- Stress Analysis of Openings in Skirts

- The Units and Design Code can be changed easily from the new Units toolbar

- Australian Wind Code

- European Wind Code

- Mexican Seismic Code

- Increased the number of input points for India's Response Spectrum Input

- Added Selection of Plate Densities for Platforms

- Added Piping Detail Input with the Mass/Equipment Detail

- Enhanced the Wind Analysis Output for several wind codes including ASCE-7 /93/95/98/02

- Added Flange Thickness Input field in the flange dialog

- Input for Additional Static Pressure

- Thick Expansion Joint Analysis for UHX based Exchanger Designs

- Sort able Nozzle Schedule Report with Weld and Pad sizes

- Plus all of the new CodeCalc features and many others

CodeCalc 2005 New Features

(by: Mandeep Singh)

The new features added to CodeCalc 2005 include:

1) ASME Ed. 2004 code updates, including UHX tubesheets, thin expansion joints (bellows kind) and the material databases.

2) Implemented the Appendix 9 Jackets in the Shell/Head module. These jackets could cover the shell (completely or part of it) and/or the head. Various types of jacket closures are also available. The program computes the required thicknesses of jacket, jacket head and closure bar. A screen shot of the jacket input is shown below:
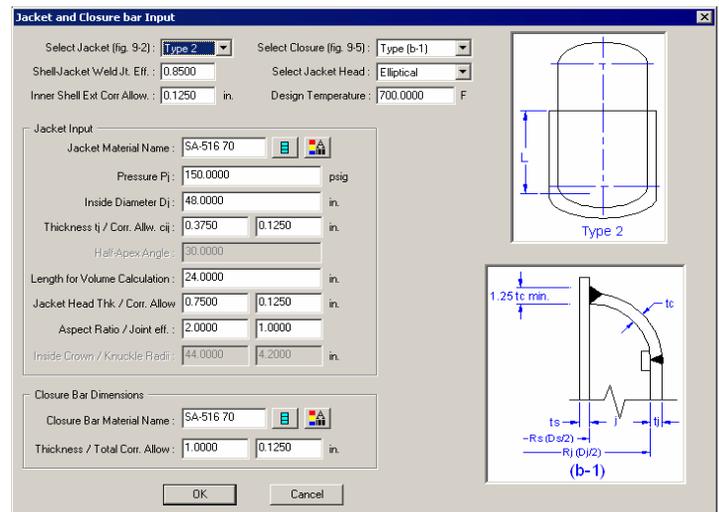
3) Incorporated thick expansion joints (flanged and flued type) into ASME UHX tubesheet module. The thick expansion joint was previously available in the TEMA tubesheet module.

4) On-screen display of error messages, warnings and important results is available for the ASME UHX and TEMA tubesheets. This speeds up the design of the tubesheet and expansion joint. See a sample below:

Tr = 1.944, Given Thk = 1.750, Maximum Average Stress Failure, Stress Ratio = 0.884 !

5) Added analysis of a vessel on 2 support lugs. Analysis for vessel supported on more than 2 lugs was previously available.

6) Added TEMA Metric, British and South African Metric bolt look up tables in flange, basering, tubesheets, floating head and Leg-Lug modules.

7) Updated Output processor.

8) Enhanced Microsoft Word generation in Output processor.

9) Enhancements were made to the user interface for the Flange module including the on-screen display of results. See the screen shot below:

Tubsh tr: 2.503   Exten tr: 0.879   Tubes ok   Shell ok   Exp Jt. Passes

10) Added an option to use the full bolt load (W*ab) in the flange design.

11) Added inputs for the partition gasket thickness and the nubbin width. In earlier versions of the program, the gasket thickness and nubbin width of the main gasket was being used for partition gasket.

12) In Annex-G module, added an option to print the membrane stress at nozzle edge.

13) Modifications to the treatment of full strength tube-tubesheet joint in the tubesheet modules.

14) Enhancements to the program for network users.

CAESAR II Version 4.50
3D HOOPS Graphics Tips

(by: Nadia Strikovski)

Starting with CAESAR II version 4.50 the 3D HOOPS Graphics has incorporated all of the features of the "standard" graphics engine and supports many new features that make it superior to the standard graphics engine. As a result the standard graphics interface has been eliminated from the Piping Input.

CAESAR II utilizes the third party HOOPS 3D graphics engine. This provides a number of benefits over the "standard" in-house designed graphics. There are a number of resources available (the COADE web site and documentation) to become familiar with the 3D graphics features (listed at the end of this article).

The HOOPS 3D graphics engine uses a real-world paradigm of a fixed model and moveable camera with the camera moving around the model as if it were attached to a string whose other end is tied to the center of the model. Zooming in and out is achieved by adjusting the string length and orbiting is accommodated by moving the camera along the outside of an imaginary bubble. The model doesn't move, but rather it is the user's viewpoint (the camera) that moves. This concept allows for the real-time interaction with the model without spending computer-processing time on recalculation of the model's position or projection.

This article is devoted to covering the most commonly asked 3D Graphics-related questions and to highlighting the most useful and/or new features. It begins with the most basic options to get the user started, continues with CAESAR II version 4.50 related tips, and ends with a troubleshooting section.

How to Display a Model in a Single Line/Center Line Mode

The initial (default) view for a new job is displayed according to the "display options" defaults (which can be set on a per job basis) that include:

• rendered (shaded 3D) view

• restraints shown

• XYZ compass shown

• isometric view

• tees and nozzles highlighted

• orthographic projection

To switch the view to display the single line mode, click the [Centerline View] toolbar button. It is located in the group of five display mode buttons [Gouraud Shading (rendered mode)/Hidden Lines/Wire Frame without hidden lines/Two Line Mode/CenterLine View] on the first row of toolbar buttons, towards the right. Alternatively, pressing the "V" key on the keyboard will toggle the views in following order: Gouraud Shading (rendered mode) -> Two Line Mode -> Center Line View.

How to Zoom the Model

There are several options to zoom the model. Interactive zooming is achieved by clicking the [Zoom] toolbar button and then moving the mouse up or down while depressing the left mouse button. The keyboard [+] and [-] keys may also be used to zoom the model in and out, once the [Zoom] toolbar button has been clicked.

The best way to zoom to a particular area of the model is to use the mouse to draw a "rubber band box" around the desired area. Simply click the [Zoom to Window] toolbar button, then left-click on one corner of the desired area, and stretch a box diagonally to the opposite corner of the area while still holding the left mouse button down. When the left button is released, the model will zoom to the selected area.

To see the entire model on the screen, click the [Zoom to Extents] toolbar button. Use the [Zoom to Extents] button instead of [Reset Plot] to view the entire model if you want to preserve all the custom highlighting and the current view.

Zoom to Window | Zoom to Extents | Interactive Zoom
Reset Plot | Zoom to Selection | Display Mode Buttons
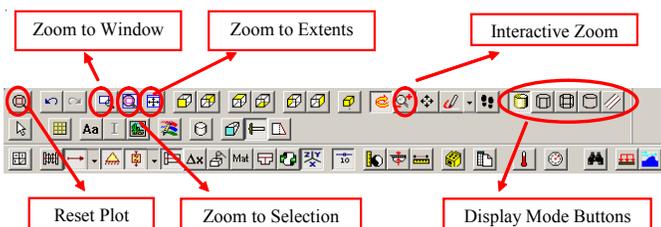
What is "Zoom to Selection?"

There are several ways to highlight/select an element in the 3D Graphics view that include clicking on an element while the [Select by Single Click] option is active, finding an element in the model with the [Find Node] option, and browsing through the spreadsheets in the Piping Input. Clicking the [Zoom to Selection] toolbar button will bring the selected element to the center of the view and zoom to it. Note: If nothing is highlighted, the [Zoom to Selection] button is disabled.

This option is especially useful together with the [Find Node] operation. The element found may be small compared to the entire model, or may be hidden behind other pipe elements or components. The [Zoom to Selection] is available directly on the Find Element dialog by enabling the corresponding check mark or from the toolbar after the Find Node operation has been performed.

Rotation/Orbiting

Interactive rotation of the model can be accomplished by clicking the [Orbit] toolbar button. Once this mode is activated, the model can be rotated by using the mouse or the arrow keys on the keyboard. To use a mouse for rotating the model, click the left mouse button on the model (the bounding box will be drawn to outline the model boundaries); while holding down the left mouse button, move the mouse around to the desired position/angle. When mouse button is released, the view will be updated and the bounding box will disappear. If the bounding box is not visible, check the corresponding box on the User Options tab of the Plot Configuration dialog (see the "3D Graphics Configuration" paragraph for details).

Note: During the rotation operation (only) for speedup purposes the model may be changed to the wireframe or single line mode view or some of the geometry details may become missing or distorted. The actual conversion will depend on the size and complexity of the model. Once the rotation is complete, the model will return to its original state.

Another way of (automatic) orbiting the model is through the **Gyro** operator. It can be activated by pressing the [**G**] keyboard key. Upon pressing the [**G**] key, the model will perform a full 360-degree rotation in the plane of view. This is useful to have a quick look around the model.

How to Rotate the Model Around X, Y, or Z Axis

Because of the concept of moving of the camera around the static model, it is not always feasible to orbit the model "about one of the global axis" (for example, preserving the vertical elements vertical) if it is in an ISO view or in any view off axis. This is because the direction cosine of the camera in an ISO view is (1,1,1), which is a little off center (the "Y-vertical model axis" is rolled forward slightly), and the camera moves around an imaginary bubble with respect to the vertical or horizontal axis of the flat screen/display.

To orbit around a true Y-axis (assuming "Y" is up), first, choose [Orbit] as your operator and set the view to the **Right** or **Left** view (either by pushing the appropriate view button or typing [**X**] or [**shift**] + [**X**]). Alternatively, the **Front** or **Back** views can be selected (by clicking buttons or typing [**Z**] or [**shift**] + [**Z**]). This will ensure that the model vertical axis is aligned with the vertical axis of the screen. Then use the "**right**" and "**left**" arrow keys on the keyboard. This will move the camera around our mythical bubble's equator, which is perpendicular to the y-axis.

Similarly, aligning any of the model's horizontal axes (**X** or **Z**) with the screen horizontal axis (by selecting an appropriate orthogonal view) and using the "**up**" and "**down**" keyboard arrow keys will move the camera around the bubble over the "prime meridian" providing the rotation about the selected model horizontal axis.

Using the Mouse Buttons and the Mouse Wheel

If the mouse is equipped with a middle wheel, it can help in reducing the time to manipulate with the model graphics.

Determine the operation desired and highlight the corresponding toolbar button. Then hold down the left mouse button and the mouse movement will cause the model to respond in the selected manner. Additionally, while holding down the middle mouse wheel any mouse movement will generate the [**Pan**] operation. Alternatively, spinning the middle mouse wheel up/down will [**Zoom**] the model in and out of the view around the mouse pointer location. Thus, using the middle mouse wheel may eliminate additional time spent clicking the [**Pan**] and [**Zoom**] toolbar buttons without actually switching the current operation.

In addition, the right mouse button can be used to display a "context menu" and an "Operations" sub menu containing toggle switches for zooming, panning, and rotating the model. The Plot Configuration dialog may also be found by right mouse clicking anywhere on the view, and selecting Properties -> Display Options.

How to Resize Restraints, Hangers

In the CAESAR II 3D graphics the size of restraints and hangers is relative to the pipe diameter. For fairly large models with long pipe runs, the size of restraints may become too small to see clearly when zoomed to the extents of the model. The size of the symbols for restraints and hangers may be manually adjusted to become larger or smaller by clicking on the black arrow to the right of the relevant toolbar button and selecting the size option from the drop down menu.

How to Change Color of Node Numbers, Pipe, Components or Background

Most of the display options can be adjusted by clicking the [**Change Display Options**] toolbar button. The various tabs that this Display Options dialog controls include: basic graphics colors, font selection and size for text data, user startup settings, and visibility (the degree of transparency). Alternatively, the same Plot Configuration dialog may be found by right mouse clicking anywhere on the view, and selecting Properties -> Display Options.

The Basic Graphics Colors Tab will be accented here: Selecting any item in the list (e.g. Nodes), then clicking the [**Change**] button, displays a standard Windows color selection tool. Selecting the desired color and clicking [**OK**] changes the color of the selected item to the new color. The rotating spring hanger is used to actively preview the color selection combinations before altering the entire plot window. This is a useful tool to prevent selecting unsatisfactory color combinations.

The colors may be reset to the CAESAR II defaults (as defined in the configuration) by clicking the [**Reset All**] button.

Note: The state of each model is maintained individually (job related), in an **XML** data file (*job-name*.xml) in the current data directory. Upon invoking another input session, CAESAR II reads this XML file and restores the 3D graphics to its previous state. This means that changing any of the colors in the CAESAR II Tools->Configure/Setup file will not immediately affect/change the current job graphics if it has already been viewed. For the settings to take effect, start the graphics, access the **Display Options** Dialog and click the [**Reset All**] button.

How to Limit the Amount of Displayed Information

If the model is large (element-wise) or has many similar looking branches, it may become difficult to review relevant information. The 3D HOOPS Graphics engine provides a number of useful tools to limit the amount of displayed information or to quickly navigate through the model.

The　[**Find Node**] option allows the user to find a specific element by providing it's "From" and "To" nodes. The "Find Node" option also allows searching for any node number and will highlight the element(s) containing this node number.

The　[**Range**] option is used to plot only those elements that contain nodes within the range specified by the user. This is particularly helpful when attempting to locate specific nodes or a group of related elements in a rather large, often symmetrical model, or when reviewing a model created by someone else.

Upon clicking the　[**Range**] toolbar button, a dialog with a sorted list of all defined node numbers with check boxes will be displayed. The user may click each individual node, select a range of nodes to display using standard Windows techniques, or use From-To node fields to add displayed nodes to the list. Upon clicking [**OK**] button, only nodes with a check mark will be displayed on the view. **Note**: The **Find Node** option may not work properly for the part of the model that is "hidden" by the **Range**. The corresponding message will also appear in the status bar.

In many cases, the elements/node numbers are not defined consecutively. Thus, it may be easier to cut/slice a piece of model at a certain location to see more details on one side of the cut. For this procedure, use the　[**Insert Cutting Plane**] toolbar button. When the cutting plane appears, use its handles to move/rotate the plane as desired. If the cutting plane's handles are not visible, or a display goes blank, the view may be zoomed too close for the plane to operate correctly. Use the　[**Zoom to Extents**] toolbar button to zoom out the model; then click the [**Cutting Plane**] button again for the handles to appear. To turn the cutting plane off, right mouse click anywhere on the view and select "**Delete Cutting Plane**" from the menu.

Annotating and Documenting the Model

It is often necessary to add a graphical representation of a model to the CAESAR II stress reports or to make screen shots for presentation. The 3D HOOPS Graphics view can be saved as a bitmap by clicking the　[**Save Image to TIF File**] toolbar button. The model geometry, colors, highlighting, as well as restraints and most of the other options will be transferred to the bitmap.

There are times when annotation is needed to clarify the model image. This could be useful to highlight problem areas, or write a brief description of the model. The annotation may be especially useful in the output processor. There are two basic kinds of annotation available: quick snap and semi-permanent.

The first type is the　[**FreeHand Markup**] that provides four options: "Free Hand", "Circle", "Rectangle", and "Annotate". It is not attached to any element in the model and is destroyed upon any manipulation to the model (like zoom or pan). It may be used to add a short description of the model to the graphics image for printing or saving to a bitmap.

The second type is the　[**Annotate Model**] toolbar button that draws a text box with a leader to an element of the model. The annotation with a leader stays with the model on zoom, pan, rotate, and use of any highlight options. It also gets printed to the printer and saved to the bitmap; however, it does not get transferred to the HTML file and is not saved upon exiting the graphics.

Another way to save an image is from **File->Save as Web Page**, or alternatively, typing the combination of [**SHIFT**] + [**H**] keyboard keys. This creates three files in the current data directory using the current job name: *.HTML, *.HSF, and *.HMF. Opening the *.HTML file displays the corresponding *.HSF file. **Note**: This is an interactive file that allows zooming, panning and orbiting. The advantage of the Web Page file is its ability to be viewed on any computer, without the need for the CAESAR II software (it requires a web browser and an internet connection to download a viewing plug-in control the first time the file is opened on that computer). This option is particularly useful to save an animated dynamics model to a file for presentation or review.

Working with Piping Input and Graphics Windows Opened

CAESAR II v4.50 allows input and edit of model data with both the Piping Input and Graphics windows opened simultaneously. This is beneficial for larger models and/or slower video cards because the graphics require some time to get re-generated every time they are opened. It also allows the user to see the model "growing" as new elements are added. If the monitor resolution allows, both input and graphics windows may be positioned side-by-side. However, in many cases the graphics window needs to be resized so small that it prevents a clear view of the model. In such cases the graphics window may still be kept open, but partially covered/hidden by the input screen.

To work in the spreadsheet with the graphics window open in the background, from the Piping Input, click the Graphics button. Then in the Graphics window click the　[**View Input Spreadsheet**] button on the toolbar to display the Piping Input

again. Clicking the same button again will toggle the Input screen on and off. If the button state is already "pushed on", then you can access the input spreadsheet by clicking the appropriate icon on the Windows task bar.

Highlight Options Available in the Input Graphics

Often it is necessary to review the piping model in the context of certain data; for example, by diameter, wall thickness, temperature or pressure. These operations are available from the 3D Graphics toolbar.

Upon clicking the corresponding toolbar button, the display is updated to show each value with a different color. A color key legend is included on the plot in a separate, resizable and moveable, window. This is a good way to verify that specific changes have been made at the appropriate locations in the model.

The first group of buttons includes Materials [Mat], Diameters [icon], Wall Thickness [icon], and Insulation Thickness [icon]. The second group includes Temperatures [icon] and Pressures [icon] with up to nine available choices. The final group will highlight and color code elements where Displacements [Δx], External Forces [icon], Uniform Loads [icon], and/or Wind/Wave Loads [icon] are defined. In the last case, the legend will be displayed in tabular format with the ability to scroll through the nine available vectors.

Note: While in the described highlighted mode, the model can still be zoomed, paned, and rotated. Any orthographic projections and single line/volume modes can still be used without affecting the model highlighted state.

Note: When the model is printed, the legend text will appear either in the upper left corner of the page or on the next page. However, the legend window/text cannot be saved to an external source (like TIFF or HTML).

Highlight Options Available in the Static Output Graphics

The Static Output 3D Graphics engine has the same general capabilities as the Piping Input Processor. Additional capabilities include showing displaced shapes in the final condition (the [icon] [**Deflected Shape**] toolbar button) and in intermediate steps (the [**Grow**] toolbar button) with the ability to set the deflection scale, highlighting and zooming to maximum displacements ([DX] [DY] [DZ] toolbar buttons), restraint loads (Forces [FX] [FY] [FZ] or Moments [MX] [MY] [MZ] toolbar buttons), and code stresses of the model ([σ] [**Overstress**], [Mσ] [**Max Code Stress**] toolbar buttons).

The major feature of the Static Output 3D Graphics is its ability to display a graphical distribution of stresses in color (a shaded contour plot) by value (the [icon] [**Stress Colors by Value**] toolbar button) and by percent (the [icon] [**Stress Colors by Percent**] toolbar button).

When using the Stress Colors options, the legend is also displayed where each color corresponds to a certain stress boundary value or percentage of the code stress. These options are used to quickly see the distribution of the code stresses in the model for a particular load case. The colors and corresponding stress levels can be manually adjusted by user.

Another useful feature of the Static Output Graphics is its ability to display any report (displacements, restraint loads, and code stresses by load cases) directly on the graphics view. Clicking the [icon] [**Show Report Viewer Grid**] toolbar button displays a "grid view" of the mentioned reports in a separate, resizable, moveable window. It has bi-directional highlight option: clicking a node or an element on the Report Viewer grid (when **Select by Single Click** mode is active) will highlight the corresponding element on the graphics view, and will zoom to the selected element if the corresponding **Zoom to Selection** mode is on; similarly, clicking an element on the graphics view will highlight the corresponding data row in the report view of the Report Viewer Dialog.

Troubleshooting

Repeating consecutive zooming, orbiting, and/or panning operations as well as using such limiting display information options like Range, may bring the model into a "difficult to understand" state. It may also be desirable to quickly remove any highlighting and other effects from the view. In such cases the [icon] [**Reset Plot**] toolbar button can be used. Upon clicking the button, the model will return to its default state. There is no need to physically close and re-open the graphics view window.

As with any software, the HOOPS 3D Graphics Engine has its internal limitations. Occasionally, the HOOPS 3D Graphics view may become corrupted after performing some operations. This may include very close zoom, using a "walk through" operator from the "top" view or backwards, among others. Since the current state of the model is maintained in the *.XML file, such problems are easily solved by simply deleting this XML file. First, close the Graphics processor window as well as the Piping Input. To delete the *.XML file, open Windows Explorer, navigate and open the data directory (where the CAESAR II input file in question is located). Find the XML data file (*job-name*.xml) and delete it. Then return to the piping input. Upon invoking the 3D Graphics engine, the model will be displayed in the CAESAR II default state and the XML file will be recreated from the configuration settings.

In some instances, the appearance of the graphics view, its ability to perform certain operations (like pan or zoom), or to display anything at all may be related to the video card in use and/or the video drivers being up-to-date (first, ensure you have the latest drivers for the video card in use). The problem here is that the 3D acceleration on many video cards is far from superior. In most cases the problems are not related to CAESAR II or HOOPS Graphics, but rather to the **OpenGL** implementation on the card. If displaying the 3D graphics causes a lot of problems, go to Tools->Configure/Setup from the CAESAR II Main Menu, select the Plot Colors tab, and remove the check mark for "HOOPS Graphics use OpenGL" to use the non-accelerated MSW drivers. Then "Exit w/Save." Upon returning to the graphics, the display problem should go away.

If zooming or rotating operations cause too much distortion to the model view, try reducing "3D Frame Rate" on the 3D Viewer setting tab of the Configuration dialog.

If all the steps above fail, check the following websites to ensure your video card isn't listed as one with known problems. If it is, the only remedy is to change the video card:

- Solidworks graphics board test results <http://www.solidworks.com/pages/services/VideoCardTesting.html>

- Techsoft (the HOOPS 3D engine developers) graphics board test results <http://developer.hoops3d.com/graphicscards/graphicscards.html>

Additional 3D HOOPS Graphics Resources:

From www.coade.com select Support->Reference Material, then select CAESAR II from the Technical Articles drop down box. The articles to review are "CAESAR II Version Usage Tips", Vol 1, Vol 2, and Vol 3.

From www.coade.com select Support->Animated Tutorials. The tutorials that cover graphics usage include "3D Graphics Usage" and "Reducer Modeling" (that also illustrates the simultaneous operation of the input spreadsheet with the input graphics).

The CAESAR II Users Guide (supplied with the installation CD or available electronically from the CAESAR\C2_Docu directory or from the CAESAR II Main Menu option Help->Online Documentation) also has detailed information and toolbar button descriptions, in relevant chapters (Piping Input, Static Output, Dynamic Output, Structural Steel Modeler) under the corresponding 3D/HOOPS Modeler Sections.

CAESAR II Version 5.00

(by: Richard Ay)

The development work on CAESAR II Version 5.00 is nearing completion. Version 5.00 will include many changes and updates, including:

- The piping input processor has been revised to provide a combined showing both the graphics and the spreadsheet. The new piping input processor has integrated the Piping Error Checker, and access to the configuration module has been provided. For debugging and problem solving, relevant sections of the spreadsheet can be *torn off* and placed on the graphics pane, allowing the spreadsheet to be hidden for larger graphics display. Additionally, sections of the model can be graphically selected and then modified.
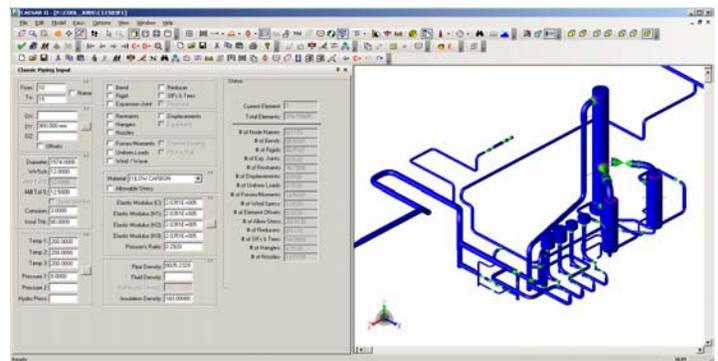
- The static output processor has been enhanced to provide users with the ability to create custom, reusable reports. The expanded static output control dialog is shown in the figure below.
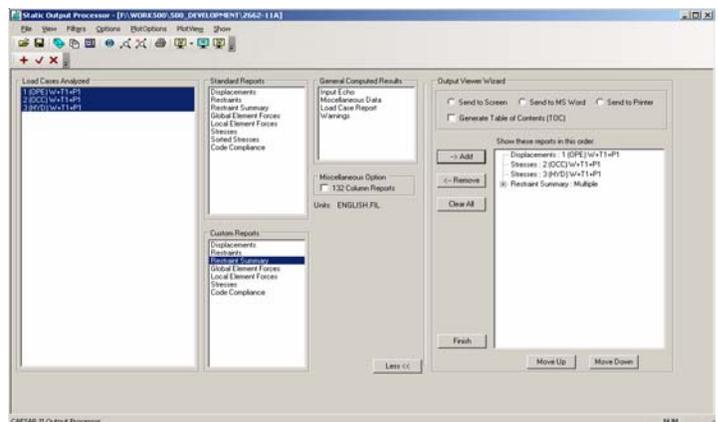
- New piping codes incorporated: EN-13480, GPTC/192, Z662 Ch 11

- Updated / revised piping codes: B31.1, B31.3, B31.8, B31.11, ASME NC, ASME ND, CODETI, TBK 5-6.

- Flexibility analysis can now be performed using the hot modulus associated with the operating temperature.

- The ability to perform dynamic analysis optionally using a *consistent mass matrix*. This obviates the need re-mesh the model for better mass distribution.

- Added Ameron Bondstrand FRP pipe specification files.

- Added PiHASA (Spain) spring hanger table.

Collaboration with CADWorx IP

(by: CADWorx Development Team)

Earlier this year, COADE Inc. released a new product that carries the ease of use and collaboration possible with CADWorx P&ID to a whole new level. The new product, CADWorx Internet Publisher (IP), allows users of CADWorx P&ID to export entire projects to a web based format that can then be shared with customers, designers, engineers, and even maintenance personnel. Collaboration, distributive design, and information sharing in plant design have been promised by software vendors for years. It has finally come of age with the introduction of CADWorx IP.

What is collaboration? When a project begins, many designers and engineers must work together and coordinate how a project or process will be constructed. Many fundamental steps must be taken to ensure complete and safe designs. Equipment must be sized and designed. Processes must be routed and instrumentation must be applied. Currently, engineers and designers perform these important steps via paper. An engineer will mark up a paper copy of the design and then have the designer or draftsman make the necessary changes in the drawings.

With CADWorx IP, the user can publish a project to a web site, a shared network server or even a CD-ROM. Other engineers, who may not have the facilities to view AutoCAD drawings, can view the drawing using only a standard browser via a DWF format. All important documentation will be available to the reviewer some of which may need to be revised. Any available reports will be automatically exported by CADWorx IP which can then be used to review line lists etc.
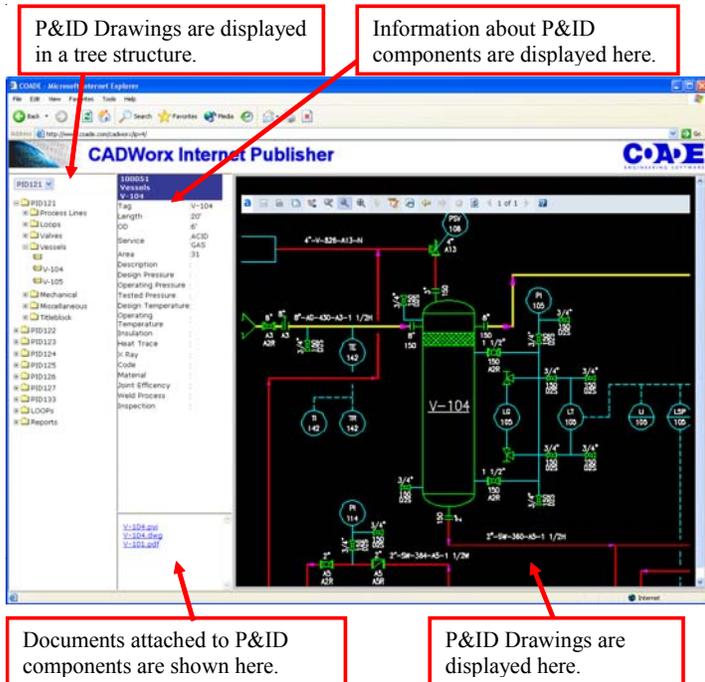
The most important benefit of CADWorx IP is the ability to give all project stakeholders (clients, subcontractors, vendors, regulators, etc.) a complete P&ID project so that they might be able to review and revise the design. Project team members can even view them from a company's internal website with the information posted daily for all concerned parties to view. Customers can also redline the design with comments and then provide it back to the engineering staff to apply the changes.

When the process/project has been completed, maintenance personnel can use the same system to maintain the process equipment. Any kind of information such as equipment specifications, catalogs, stress analysis reports etc. can be stored with a CADWorx P&ID project and thus any type of information can be published with CADWorx IP. For example, if the maintenance personnel change something for required regulatory requirements, it can be redlined for engineering to make the necessary modification to the drawings. All that is required is a computer with Internet Explorer and one of Autodesk's DWF viewers.

The export process is straight forward and easy. The user can start by selecting the project configuration file for the P&ID project they wish to export. The entire export process is completely customizable with the user having complete control on what is exported. Just following a standard windows wizard, they can select which drawings, tables, columns are to be exported. CADWorx IP can even export any reports the user created in the project database. If any loop drawings were created, those too are available for export.

CADWorx IP supports several viewers available from Autodesk, Inc. Some of these are available as a free download while others require license. More information on the kinds of available viewers is supplied with the program.

CADWorx IP shares the CADWorx philosophy of keeping things simple and the evidence is clearly visible with the final product. The exported P&ID drawings are laid out in an easy to use and intuitive manner allowing for quick access to all information accompanying the drawing as shown in the illustration below.

P&ID Drawings are displayed in a tree structure.

Information about P&ID components are displayed here.

Documents attached to P&ID components are shown here.

P&ID Drawings are displayed here.

The easy to use panel display and its integration with Internet Explorer ensure that virtually no training is required for someone to use CADWorx IP. On the left panel, a tree display shows the various P&ID drawings, loop drawings and reports that were exported. The upper middle panel shows all the information associated with each P&ID component. Any documents attached to the project are also automatically available through the lower middle panel. And of course, the drawing itself is available complete with zoom/pan controls etc. View a complete sample at http://www.coade.com/cadworx/ipv4.

CADWorx IP is available now! It is included at no additional charge in the CADWorx P&ID Professional package. If you would like to more information about this product, please contact your local dealer or sales@coade.com.

COADE's Plant Design Strategy: Seeking the Integrated Total Solution

(by: Tom Van Laan)

In 1996, COADE introduced CADWorx Pipe, our AutoCAD-based plant design application to our roster of products – a roster that up to that point had included only Plant Engineering (analytical) applications. Since that time, we have continued to expand our plant design offerings (which today include CADWorx Plant, P&ID, Equipment, Steel, and Internet Publisher) — all the time with our eye firmly on a goal of which many of our customers may not be aware.

The addition of plant design solutions to our existing Plant Engineering solutions probably gave the impression that we were moving toward offering a 'total solution'; one that encompassed all aspects of plant design and engineering. On the surface, that is true, but our true strategy has really been so much more than that.

The 'Total Solution' is Born

In the past most companies and end users asked for a tool that would aid them in designing and constructing the entire project. We in the plant design software industry developed a catch phrase for this elusive Holy Grail — the 'total solution'. From the inception of the first plant design system the goal of the developers was to deliver that 'total solution'. But the question was how? There were many possible routes to reaching that goal.

Delivering a 'Total Solution'

The challenge facing all developers was not only to deliver a 'Total Solution' that was marketable (and believable) on that basis, but also one that not only met the needs of the organization as a whole, but also those of the individual disciplines. Means of achieving such a solution included:

- Recommendation
- Partnering
- Re-branding
- Acquisition
- Development

Recommendation

This is the weakest way to provide a component of the total solution – the developer simply points a user in the direction of a supplier who can provide what is needed. In this case any interaction between the developer's system and the application being recommended is generally on a very basic, and perhaps coincidental, level.

Partnering

This method is slightly stronger than recommending a system, and usually involves a situation where the developer's system and the complementary application may have undergone some optimization to facilitate interfacing between the two, and where the complementary application may even be resold or supported by the developer. Typically the partnering arrangement does not affect the look and feel of either solution but may provide for the interaction features deemed most important.

Re-branding

This solution usually gives the end-user a nice fuzzy feeling that they have purchased an application from the same developer that has done such a good job providing the rest of their system, but in reality it is usually no better than partnering. In most cases the methodology, workflow and command structure is still at variance to that of other the other components of the 'Total Solution' supplied to the end-user. Re-branding is simply a *marketed* form of *partnering*.

Acquisition

This shows commitment on the part of the developer, not only a financial commitment, but also a commitment to take on the not inconsequential development and marketing roles for the product. The aim here is to own the technology for the long term, with plans to integrate it into the developer's suite of software. The plus is that this provides the tools the market requires much more quickly, in effect by buying knowledge that would otherwise take much longer to develop internally. The minus is that in the short term the applications may not have the look and feel of the existing offerings from the developer.

Development

This is the best of the best. The benefit is that users get a product developed by their chosen developer that has the look, feel, and interactivity that they expect. The minus is development from scratch takes a lot of time, resources, and money to pull off – in fact a fortune can be spent, especially over the time that it may take the developer to play 'catch-up' with other software vendors.

Where the Total Solution fails to deliver

In General the 'total solution' delivered by developers in this industry comprises a combination of all of the above strategies. This is unavoidable but what we have seen being offered to the market in many cases is no more than a bag of parts with minimal interface between those parts – in other words, no more than a way to 'check the boxes' on the brochure indicating the solution's capabilities.

After speaking to end users and brainstorming internally, we came to the conclusion that this is not what was really wanted. Yes, end users wanted it all, but they also wanted the parts to work as a whole and to interface at their most natural level.

Therefore what the market was asking for was not the 'total solution' that developers were giving them — but what they really wanted was an *integrated total solution*.

Moving Towards Integration

As software applications mature they acquire a user base that is comfortable with the workings and performance and those applications. For true integration the trick is to take what users already do and what they already produce and make that same information accessible to other interested parties involved in the work process.

We can therefore differentiate the 'total solution' as opposed to the 'integrated total solution' thus:

A Total Solution is a solution that has all the tools required to complete a task or project.

An Integrated Total Solution is a solution that not only has all the tools required to complete a task or project, but also a solution where information is shared between all of those tools to the extent that the benefits of one project team member's work can be accessed by all other project team members.

The integrated total solution exemplifies COADE's vision. We believe that our future, and the interests of the end users of our products, is best served by concentrating on integration. This is not a new focus but one that has steadily become part of the culture of the company.

Only One Way to Get There?

Although developing applications in house gives the best content control and therefore the best chance of making integration easier, this does not always make it the best decision.

At some point in the process a developer has to admit that what they can bring to the market may not be the best solution, or it may not be accepted widely enough, for it to be a viable product to bring to market. In those cases honesty dictates that partnerships, re-branding, or acquisition are better strategies all round. An example of this is seen in COADE's relationship with NavisWorks, whose products (Roamer, Presenter, Publisher, and Clash) have been integrated with CADWorx Plant to perform the design review, presentation graphics, and clash detection functions for the COADE plant design suite.

Integration in practice

COADE's initial foray into integration resulted in the link between our CAESAR II pipe stress program and our CADWorx Plant design suite. These two applications were naturals for integration as the work done in both design and analysis has such a direct and immediate impact on each other. What made this integration special (easier) was the fact that as both applications are developed by COADE, the developers of each program could work very closely (both figuratively and literally) together. The success of this integration, which allows the stress engineer and piping designer to work exactly as they always have, but now they can benefit directly, in as transparent a form as possible, from the work done by their counterpart across the departmental divide – encouraged us to work toward a completely integrated solution.

COADE's Integration Path and Vision

This drive for integration did not stop at the obvious link between CADWorx Plant and CAESAR II. We have continued (and will continue) to pursue:

- PVElite and CADWorx Equipment

   CADWorx Equipment users can now seamlessly import Pressure Vessel and Exchanger designs that have been designed with PVElite. This link will be extended to a bi-directional link in the first quarter 2005.

- CADWorx Plant and CADWorx P&ID

   Intelligent information from CADWorx P&ID projects can be read by CADWorx Plant to set parameters for any line that needs to be modeled. Line number, line size, specification, service, area, unit etc. can all be set automatically in CADWorx Plant simply by selecting a line from CADWorx P&ID.

- *CADWorx Internet Publisher and CADWorx P&ID*

   CADWorx Internet Publisher can publish all data (including linked documents) from a CADWorx P&ID project to the internet, intranet, or CD and allow access to that information to anybody with any standard browser, regardless of lack of CAD platform, applications, or skills.

- *CADWorx Plant and CAESAR II*

   At the moment CADWorx Plant and CAESAR II can bi-directionally transfer models and other information between the two programs.

- *CADWorx Steel Professional and CAESAR II*

   The first quarter 2005 will see the release of CADWorx Steel Professional. This new application will have enhanced capabilities over CADWorx Steel including interfaces with CAESAR II, ROBOT, and other structural analysis products.

- *CADWorx Plant or P&ID and Access, SQL Server, or Oracle*

   CADWorx Plant seamlessly links to any ODBC compatible database to produce bills of materials or reports (line lists, etc.) available to any application that supports similar standard databases.

- *CADWorx Plant and Isogen*

   CADWorx Plant interfaces with Alias' industry standard software to produce automatic isometrics.

- *CADWorx Loops and CADWorx P&ID*

   CADWorx Loops produces automatic instrument loop diagrams from the CADWorx P&ID project database.

Quantifiable Benefits of Integration

Obvious benefits of an integrated solution are evident in the fact that data reuse leads to work hour savings and reduced mistakes incurred during manual or semi-automatic data transfer. In fact one study (by Construction Excellence, www.m4i.org.uk) makes the claim that over 80% of inputs are wastefully repeated due to lack of integration. Another study reports that the annual cost of rectifying construction defects caused by errors in transferring information to detailed drawings (and by operatives subsequently being given incorrect instructions) approaches to $2 billon in the UK alone.

This indicates that one of the best ways to save costs is not by investing in faster machines or increasing the efficiency at each of the engineering, procurement, or construction stages of the process but by making sure information input is made useful at each subsequent stage of the process.

An Integrated System, a Distributed System

A major benefit of a system that shares all information is that it allows work flow tasks to be broken up and distributed to different project team members – this may be done on a regional or discipline level, between EPC and owner/operator, or in many other ways. If all team members can see and benefit from the work of all others, there is no need to face the choice of either having to assign the entire project design to a single individual (or single group) or risk clashes and other incompatibilities between different team members.

This shows that distributive design may well be the true payoff for an integrated engineering system.

Distributive Design the Future

So what is the end game? What does COADE feel end-users will be expecting (and demanding)?

First – the end game is that that we must cut the loss of data from one stage of the design build process to another. This should be a main focus of any engineering, procurement or construction company looking for the best efficiencies.

Secondly – Developers should not live in a bubble. We owe it to the market to increase the integration between the applications that we develop AND to link to industry standard packages supplied by other software developers.

Third – Look not only at what a package can do for one project team member but also what benefits the application's integration can deliver to other team members both at the same stage in the process and for those downstream.

Your call

We hope that 'check box' features and benefits will not be the only criteria upon which you select your plant design solution and that integration and distributive design will now also be criteria considered on your path to implementing your ideal plant design system. For those who do, we thank you in advance for putting your faith in COADE's product line – your first step to true integration and design efficiency.

TANK 2.50 and New Bolting Procedure

(by: Richard Ay)

With the publication of Addendum 3 to the 10th Edition of API-650, API has significantly changed the bolting requirements for tanks. Prior to this addendum, bolting requirements were mentioned in a number of places in API-650:

- Section 3.11 for overturning stability defined a bolt load
- Appendix E for seismic evaluation defined a bolt load
- Appendix F for internal pressure evaluation provided an allowable bolt stress, but offered no other guidance.

With the publication of Addendum 3, the API-650 code has replaced the above sections with a single statement that instructs the engineer to refer to Section 3.12 (a new section) for bolting guidelines. Section 3.12 provides minimum requirements for anchorage, through Table 3-12 which lists 7 loading conditions. For each loading condition, the code provides and equation for the net uplift as well as the allowable anchor bolt stress. The 7 loading conditions to be addressed are:

- Design pressure
- Test pressure
- Failure pressure
- Wind load
- Seismic load
- Design pressure + Wind
- Design pressure + Seismic

These new bolting requirements are addressed by TANK Version 2.50. TANK will evaluate these 7 conditions, when applicable, and provide a concise bolting report to the user. A typical bolting report is shown in the figure below.

This report shows that only the seismic conditions are applicable to this particular tank, and as a result, 16 bolts, 2-1/2 inches in diameter are required. (The "zero" values in the "uplift" column indicate that either no load was defined, i.e. pressure, or the load was too small to produce uplift, i.e. wind.)

Users who have not acquired Addendum 3 should do so as soon as possible. If you are not using Addendum 3, your tank bolting does not meet the requirements of the current API-650 code.

This change in Addendum 3 means that a tank designed to a previous edition of the code may now require bolts, whereas before it didn't. Users need to be wary of code changes. The same model run with TANK Version 2.40 may yield different results than when run with TANK Version 2.50. Because of this "code change", enough design information must be provided to TANK, in case bolts must be designed. The input dialog for bolting information is shown in the figure below.
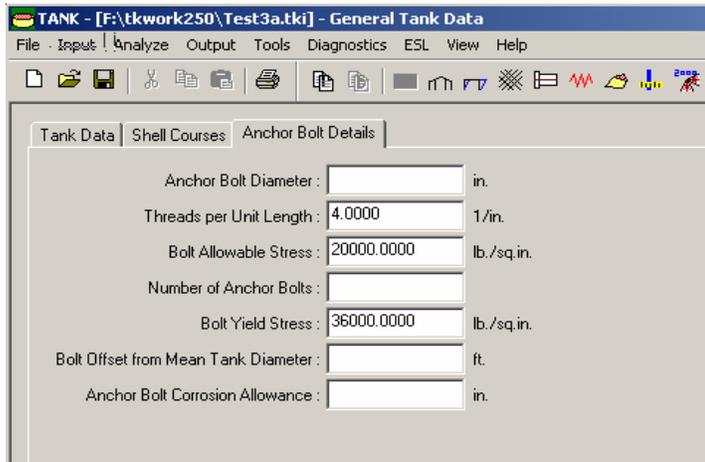
This dialog shows the minimum amount of information that is necessary for an analysis. Even if you think your tank doesn't need anchorage, this data is required. An increase in wind speed could always change the outcome, requiring anchorage.

Stress Analysis of Skirt Supports with Openings

(by: Ray Delaforce)

Process towers are usually supported on skirts. Often, because of the bottom drain outlet, for venting (for example hydrogen) it is necessary to have openings or nozzles in the skirt. This is especially true in the case of access openings that are fairly large in diameter. Cutting holes in skirts has a weakening effect. The weakening is caused by the reduction in the section properties where the cross-sectional area and section modulus are reduced at any horizontal cutting plane passing through the opening.

It has been a common practice for engineers to simply replace the area removed by attaching a reinforcing pad or frame around the opening. However the actual problem is more complex than just adding material for area replacement.

Consider the following situation. Shown below in figure 1 is a skirt with two openings.
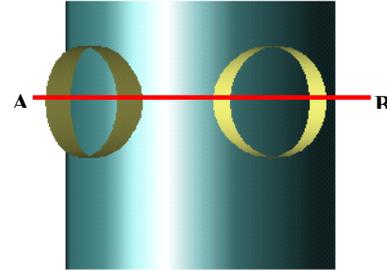

Figure 1

At the cutting plane A-B metal has been removed. However, let us first consider the case of a skirt without any openings shown in figure 2. In this case, the section properties are given by:

$$A = \pi r^2$$

$$I_{xx} = \frac{\pi D^3}{8} \qquad Z_{xx} = \frac{\pi D^2}{4}$$

$$I_{yy} = \frac{\pi D^3}{8} \qquad Z_{yy} = \frac{\pi D^2}{4}$$

Where:
D   is the mean diameter
t   is the skirt thickness
I_{xx} is the second moment of area about X-X
I_{yy} is the second moment of area about Y-Y
Z_{xx} is the section modulus about X-X
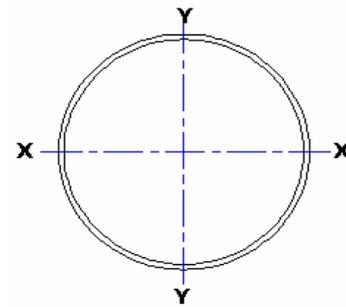Z_{yy} is the section modulus about Y-Y

Figure 2

Notice that the value of I_{xx} and I_{xx} are equal, and that the center of area – or centroid – passes through the center of the circle in figure 2.

Once we cut holes in the skirt the situation becomes more complex. In figure 3, we see a skirt with two access opening:

The centroid moves to a new position as indicated by the checkered circle. The next task is to find the principle axes. For a non symmetrical section, the principle axes Y'Y' and X'X' rotate to a new position, passing through the new centroid. This task is accomplished in PVElite by performing a numerical integration. The new values of Z_{xx}' and Z_{yy}' are then computed, the lesser of which is the new section modulus.
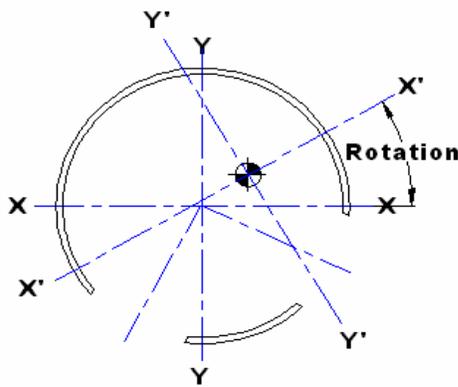

Figure 3

Sloping Skirts with obround access openings:

Now consider the case of a sloping skirt, with obround access openings. In this case the situation is different. We can use the same method to determine the section properties through any horizontal plane. Looking at figure 4 this is what we mean:

The cutting plane A-B now passes through several openings. As the distance of the cutting plane from the bottom of the skirt increases, the skirt diameter decreases.

Now, PVElite has the task of finding the location of the cutting plane such that it returns the weakest section properties. PVElite searches for the location of the smallest value of Z_{xx}' or Z_{yy}' whichever is the lesser.
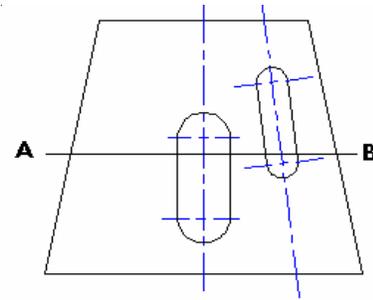

Figure 4

Performing the computation manually is extremely tedious, but as the computer does not tire out, it is the ideal platform for this task.

Before we look at an example of a particular skirt configuration, let us digress for a moment to look at how pressure vessel codes handle this situation.

The German AD 2000, and European EN 13445 codes provide a procedure for determining the weakening effect of a single opening in the skirt. The computation to determine the section properties at the widest part of the opening is not given in the code, but this is not very difficult to compute by hand (although still tedious). As far as we are aware, no code addresses the situation where there are multiple obround openings in a sloping skirt. As this situation arises from time to time, we have included this analysis in PVElite as an optional method available to the user.

Let us now consider an example for a tower with a tapered skirt with obround openings located at various heights in the skirt. The input screen for the openings looks like this:
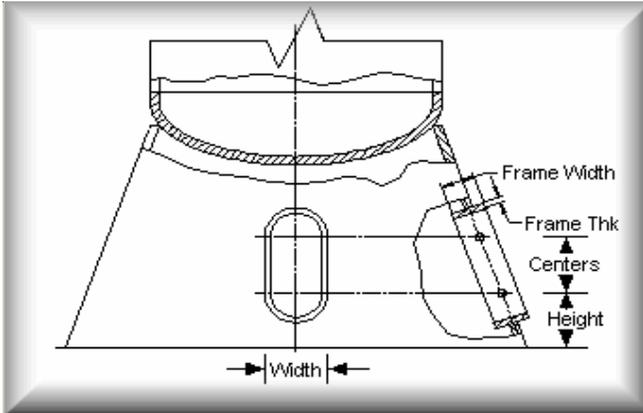

This indicates there are three openings in the skirt. The diagram that goes with this dialog is shown below.

The three openings are located at 0°, 110° and 230° around the skirt at varying heights.

We have created a tower with a tapered skirt, with a wind load that subjects the skirt to a bending moment and shear force.

Without going into the details of the tower and wind forces, we can examine the output generated by PVElite for this skirt. Notice that the program computes three sets of section properties namely:

- At the position where the user has entered a height value (1000 mm) see above

- At the location of the least section modulus (the weakest section)

- At a location that give the least cross-sectional area.

The program computes the bending stress at the weakest section (at the least section modulus Z). It also checks the ligaments between the openings to ensure there is no interference between adjacent openings.

This is a sample of the computed output:

Skirt Support Analysis Generally per AD 2000 and EN 13445-3: 2002:

Item: Node 10 to 20

Note: All dimensions at the cutting height are at the Mean Diameter and, the whole skirt must be ONE piece.

```
This is a Conical type Support Skirt
Wider at the Bottom
Half Apex Angle of the skirt        a        1.432  Deg
Skirt Mean Diameter  Bottom End  DzBot  2169.056  mm.
Skirt Mean Diameter  Top    End  DzTop  2019.056  mm.
Skirt Analysis Thickness (Corr.)   ez       3.175  mm.
Skirt Height                       H    3000.000  mm.
Applied Moment - Top    of Skirt  M4t  28335248.000  N.mm.
Applied Moment - Bottom of Skirt  M4b  59060612.000  N.mm.
Weight Above the Skirt            F4    33031.203  N.
```
Skirt Details

Access Opening: 2 Cuts through the bottom of the skirt

Access Opening: 3 Cuts through the bottom of the skirt

Note: If the opening cuts top or bottom, this is not a problem
This becomes a 'Sentry Box' type Opening

```
                    Layout  Cntr. Dist.  Obround
Access No  Diameter  Angle  From Bottom  Cntr-Cntr
             mm.     Deg.      mm.        mm.

    1      39.3701   0.0000   23.6220     7.8740
    2      35.4331  110.0000  11.8110    19.6850
    3      37.4016  230.0000  15.7480     5.9055
```
Opening Details

```
Access Opening Frame Dimension
Access No    Width    Thick
             mm.      mm.

    1       1.9685   0.3937
    2       1.9685   0.3937
    3       1.9685   0.3937
```

```
Interference check:
Access     Acess     Pitch      Ligament
                      mm.         mm.

    1        2      2043.4139   1058.6173
    2        3      2249.5925   1303.5631
    1        3      2043.4139   1058.6173
```
Interference Check

```
INTERFERENCE CHECK:  No Interference Detected between Openings

 The Minimum Pitch between openings    is:  2043.4139 mm.
 The Minimum Ligament between Openings is:  1058.6173 mm.
```

Section properties for the user input cut height [User]

```
Number of Openings At Critical Height = 3

Access No        Cut Angle 1        Cut Angle 2
                   Deg.               Deg.

    1            -25.6270           25.6270
    2             87.6375          132.3625
    3            221.7481          238.2519
```
Where the openings cut the Skirt

```
The points for integrating Ixx, Iyy is as follows:
FROM THETA1     TO THETA2
   Deg            Deg

   25.627        87.638
  132.362       221.748
 -121.748       -25.627
```

```
Hght of the Section Cutting Plane  H         1000.000  mm.
Diameter at the Cutting Plane      d         2119.056  mm.
Rotation of Principle Axes         a          -24.282  Deg.
Centroid Position in X direction   g          -88.510  mm.
Centroid Position in Y direction   h          -93.965  mm.
Principle Moment of Intertia      Ixx  10297201664.000  mm**4
Principle Moment of Intertia      Iyy   9088164864.000  mm**4
Distance to Extreme Fibre        ybar       1183.165  mm.
Distance to Extreme Fibre        xbar       1103.155  mm.
Minimum Section Modulus            W4    8238349.500  mm.^3
Metal Area at the Cutting Plane    A4      17536.994  mm²
```

Section properties at weakest section modulus [Z]

```
Number of Openings At Critical Height = 3

Access No          Cut Angle 1          Cut Angle 2
                      Deg.                 Deg.
_____

    1               -27.8876             27.8876
    2                85.1045            134.8955
    3               203.9478            256.0522
```

Weakest Section

```
The points for integrating Ixx, Iyy is as follows:
FROM THETA1      TO THETA2
    Deg             Deg
_____

  27.888          85.104
 134.896         203.948
-103.948         -27.888
```

```
Hght of the Section Cutting Plane  H         622.193   mm.
Diameter at the Cutting Plane      d        2137.946   mm.
Rotation of Principle Axes         a          31.694   Deg.
Centroid Position in X direction   g         -20.349   mm.
Centroid Position in Y direction   h         -16.930   mm.
Principle Moment of Intertia      Ixx   9056479232.000 mm**4
Principle Moment of Intertia      Iyy   8047504384.000 mm**4
Distance to Extreme Fibre        ybar       1074.275   mm.
Distance to Extreme Fibre        xbar       1096.770   mm.
Minimum Section Modulus           W4      7337467.000   mm.^3
Metal Area at the Cutting Plane   A4        14988.990   mm²
```

Least Section Modulus Z

Section properties at least metal area [A]

```
Number of Openings At Critical Height = 3

Access No          Cut Angle 1          Cut Angle 2
                      Deg.                 Deg.
_____

    1               -27.8622             27.8622
    2                85.1233            134.8767
    3               203.7489            256.2512
```

```
The points for integrating Ixx, Iyy is as follows:
FROM THETA1      TO THETA2
    Deg             Deg
_____

  27.862          85.123
 134.877         203.749
-103.749         -27.862
```

```
Hght of the Section Cutting Plane  H         591.895   mm.
Diameter at the Cutting Plane      d        2139.461   mm.
Rotation of Principle Axes         a          32.103   Deg.
Centroid Position in X direction   g         -19.196   mm.
Centroid Position in Y direction   h         -15.586   mm.
Principle Moment of Intertia      Ixx   9073734656.000 mm**4
Principle Moment of Intertia      Iyy   8044569600.000 mm**4
Distance to Extreme Fibre        ybar       1074.319   mm.
Distance to Extreme Fibre        xbar       1095.862   mm.
Minimum Section Modulus           W4      7340867.000   mm.^3
Metal Area at the Cutting Plane   A4        14979.139   mm²
```

Analysis of the Section at Weakest Section Modulus [Z min]:

Please Note: The following calculations only concern the minimum. Modulus [Z]. Section properties for the minimum area [A], and the user cut height [User] are not included below

```
Find the moment acting at the Critical Height [M4]:
 = (Mbottom - Mtop)*(H - CutHeight)/H + Mtop
 = (59060612 -28335248 )*(3000.000 -7466.313 )/36000.000 + 28335248
 = 52688248 N.mm.

Radius of Gyration of Skirt at 4-4 [r]:
 = Sqrt( Min( Ixx, Iyy ) / Area )
 = Min( 21758.301 , .80475E+10 ) / 14988.990 )
 = 732.731 mm.
```

```
Skirt Slenderness Ratio [r/L]:
 = 732.731 / 36000.000 = 0.244

Location of the Skirt Centroid at critical height [e]:
 ex = -20.349 : ey = -16.930
  e = Sqrt(ex² + ey²) = Sqrt(-20.349 ²+-16.930 ²)
    = 26.471 mm.
```

New section Centroid

```
Moment Increment at Centroid [DeltaM]:
 = Weight(F4) * Eccentricity (e)
 = 33031.203 * 26.471 = 10496672 N.mm.
```

Extra Moment resulting from movement of centroid

```
Axial Stress at Residual (Weakest) Surface Area [Sigma]:
 Downwind Side:
 = -( M4 + DeltaM ) / W4 - F4 / A4
 = -( 52688248 + 10496672 ) / 11373.097 - 33031.203 / 14988.990
 = 5.921 N./mm²

  Upwind Side:
 = +( M4 + DeltaM ) / W4 - F4 / A4
 = +( 52688248 + 10496672 ) / 7337467 - 33031.203 / 14988.990
 = 10.31 N./mm²
```

Bending Stresses

```
Maximum Allowable Compressive Stress [SC]:
 = From External Pressure Method = 20.000 N./mm²
```

This value will depend upon the code employed

```
Maximum Allowable Tensile Stress [ST]:
 = Min ( 1.2 * Design Stress , 0.90 * Yield Stress )
 = Min ( 1.2 * 137.900 , 0.90 * 239.946 ) = 165.480 N./mm²
```

SUMMARY:

Actual Compressive N./mm²	Tensile N./mm²	Allowable Compressive N./mm²	Tensile N./mm²	Ratio Comp.	Tens.
5.921	10.31	31.30	165.480	0.189	0.062

Results are satisfactory

Please note the following:

The Minimum Ligament between Openings = 1058.617 mm.
The user must assure that the stresses in the ligament area are acceptable.

PVElite 2005 ©1993-2005 by COADE Engineering Software

Specifying Hillside Nozzles in PVElite

(by: Ray Delaforce)

Many users have asked us about the 'Layout' button on the nozzle dialog box. This is the button in question:

For some reason there has been confusion as to how this works, especially in the case of hillside nozzles. So let us start from scratch.
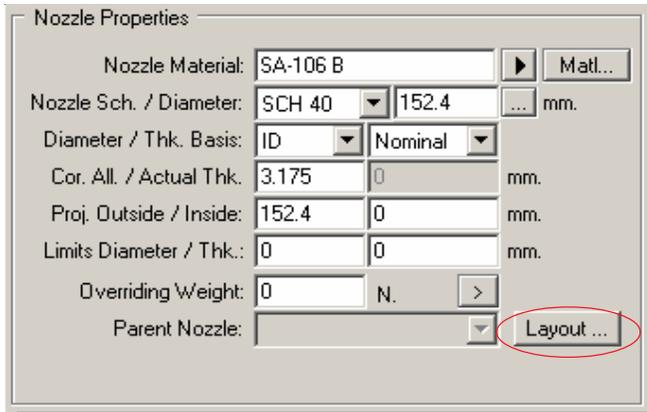
Figure 1

Consider a vertical vessel, and let us look at the cylindrical shells. Looking from the **top**, the angular positions around the shell are as shown here in figure 2:
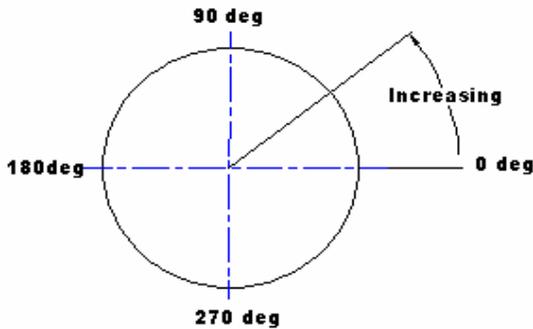

Figure 2

The angle around the shell increases in the counter clockwise direction. Let us now consider the case of a hillside nozzle. Once we have pressed the Layout button, we get the screen shown in figure 3 below. Click the 'Hillside Nozzle' radio button as shown.
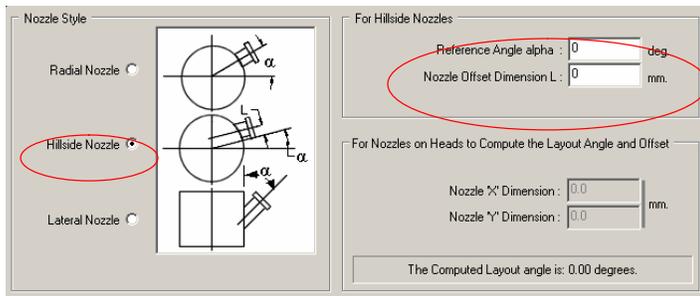

Figure 3

Reference Angle and Offset:

Reference angle: This is the DIRECTION in which the nozzle POINTS. This is a very important point. In figure 4 note that the angle is where the nozzle is pointing. The nozzle can be anywhere in space, but only its **direction** is the reference angle. The datum from which the reference angle is taken is the 0 degree point on the shell. The angle in which the nozzle points can be anywhere from 0 deg to 360 deg. The angle is measured counter clockwise from the 0 degree datum point.
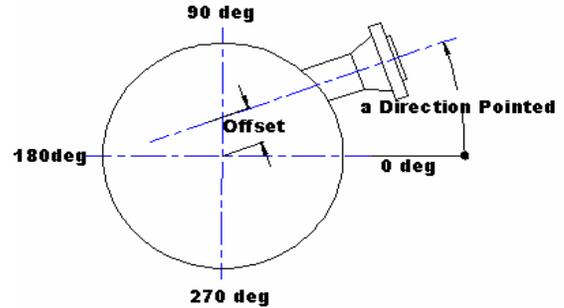
Figure 4

The offset distance is the distance from the centerline (or axis) of the nozzle to the centerline (or axis) of the shell. Now, the offset is also considered in the counter clockwise. In figure 4, the nozzle centerline is above the shell centerline. This is what is meant by the fact that it is also counter clockwise. If the nozzle were below the shell axis, the offset dimension would be a negative number.

Layout Angle

This is the angle at which the centerline of the nozzle cuts through the shell wall. This is illustrated in figure 5:
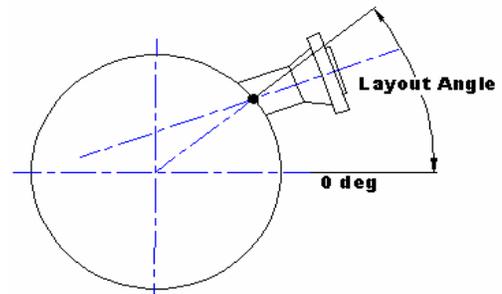

Figure 5

The layout angle is computed by PVElite once it has been given the reference angle and the offset distance. This value is important for placement and interference determination.

The computed layout angle will be found on the main nozzle dialog screen (computed from the reference angle and offset distance. This is shown in figure 6.
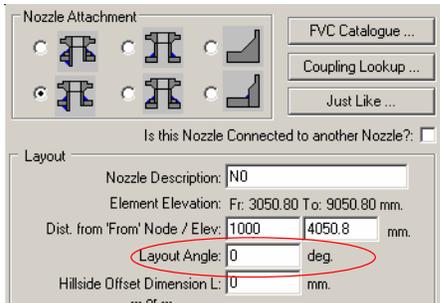

Figure 6

Hillside Nozzles on Heads

In the case of heads, the same principle applies as in the case of hillside nozzles in a cylindrical shell. However, a hillside nozzle installed in a head can be entered in two different ways. Firstly, figure 7 illustrates hillside nozzles in a head:

The axes of the nozzles are all parallel to the axis of the head.
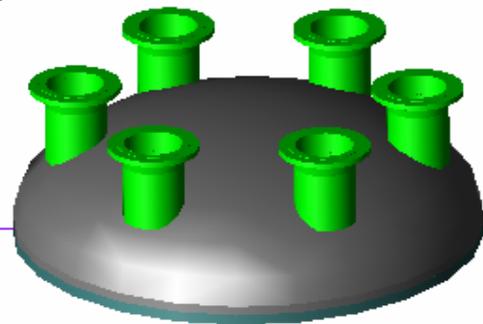
Figure 7

If the nozzle is being installed in a head, pressing the 'Layout' button will reveal this dialog screen as illustrated in figure 8:
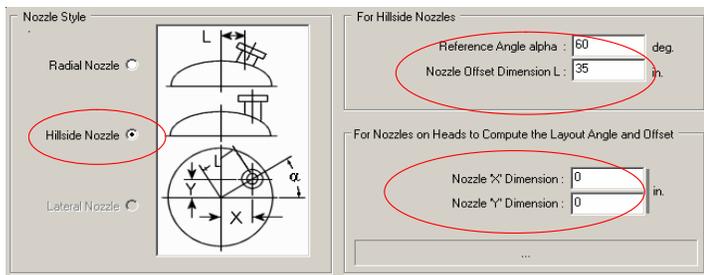

Figure 8

There are two methods of locating a hillside nozzle on a head. The first is to give the reference angle and distance from the center of the head – the offset. Here are the two dimensions as shown in figure 9a and 9b:
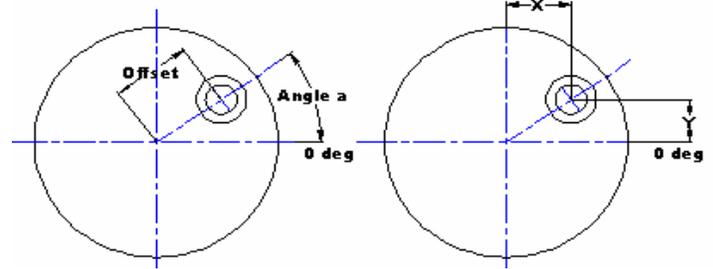

Figure 9a   **Figure 9b**

Figure 9b shows how the location of the nozzle can be specified in Cartesian coordinates by giving the X and Y dimensions. This is very useful as nozzles are often specified in this manner. In this case the offset and reference angle are calculated by PVElite.
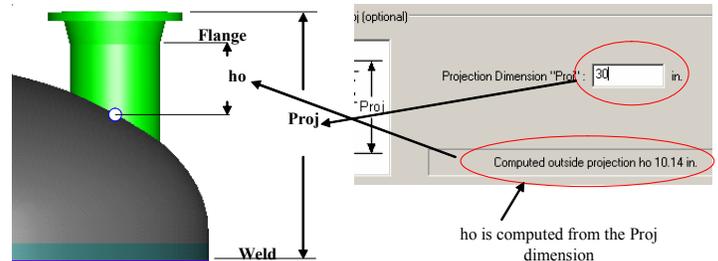
Nozzle Projection

Look at figure 10.

Figure 10

Enter the dimension 'Proj' from the head weld line to the face of the flange, and PVElite will compute the dimension 'ho', the distance from the where the centerline of the nozzle intersects the inner surface of the head to the end of the nozzle where it is welded to the nozzle flange. When you close the layout dialog screen, the nozzle dialog screen will show the computed value of 'ho' as shown in figure 11.

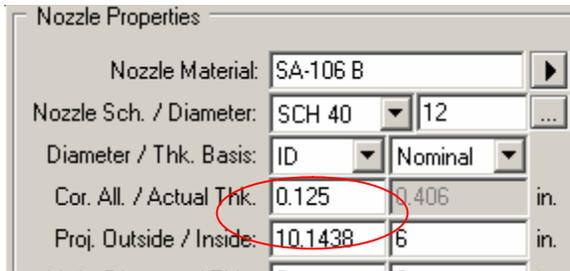In this way, the tedium of computing the projection of the nozzle is relieved.

Figure 11

Conclusion

The purpose of the layout button on the nozzle dialog screen is to simplify the placement of hillside nozzles. PVElite performs all the computations necessary for this task without the user having to be concerned about the angle of the nozzle relative to the component surface, or the elongation of the hole in the component. PVElite does it all.

What is ODBC?

(by: John C. Luf, Consultant, Cleveland, OH)

CAESAR II has had the ability to write model data out to ACCESS™ since V4.2. Many users probably never bothered to go through the cumbersome set up required to utilize this feature. V4.5 automatically sets up the PC it is being installed on for this feature.

ODBC is the acronym for *Open Data Base Connectivity* (a Microsoft development). The goal of ODBC is to make it possible to access any data from any application, regardless of which database management system (DBMS) is handling the data. So as you can see this is a standard across the Microsoft Empire.

A pipe flex model is nothing more than data and its results are also data, so the ODBC is a tool for handling data.

How does it work?

The use of the ODBC transfer is handled via the data base wizard found under the tools section of the menu bar. A review of the technical reference manual under the interfaces section will show how to use the ODBC transfer. (Note you do not have to set up a DSN as mentioned in the technical reference the installation process should have already accomplished this for you.

After review of the technical reference manual you will be able to write data via the ODBC interface.

So why bother?

Why trouble oneself with this? Admittedly I agree CAESAR II has a very rich set of report writing options in place for input and output data that do not involve ODBC. However some uses that I have found handy are… Specialized custom restraint summaries to other parties, extracting input data for model review…
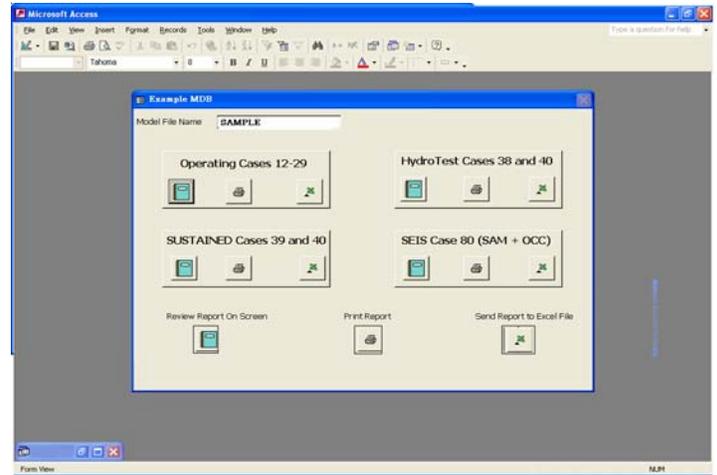
Input review…

One of the things that always is handy is to know what the block structure of a model is like. In the old days (15-20 yrs ago) model analysis was a big deal and models were constructed and maintained with religious zeal manually. These days models may be automatically extracted from 3D models, models or parts of models may be shared in collaborative engineering efforts, and nowadays lines which have been analyzed are just as much subject to revision and maybe more so than lines which have not been analyzed. What this means is that seldom do models progress from node 10 to node 10000 in a smooth logical sequence. When trying to edit such a file lack of knowledge of the block structure can add to your headaches.

What's a block and what's block structure???? An example is listed in the table below:

From	To	Comment	Block 1
10	20	Start of File, from Pump1 Discharge	
20	30		
30	40		
40	50		
50	60	End at tee/ Header	
70	80	Bypass around Control Valve	Block 2
80	90		
90	100	End Bypass	
150	160	Etc. Etc.	Start of
160	170		Block 3

Simply extracting the element data via the ODBC and then exporting it to Excel™ (very simple to do within ACCESS) the from and to node data then writing a formula to compare node numbers will allow you to easily find nodal breaks and define the files block structure.

"Splash Screen ACESS Switchboard form"

"Custom Report"

There are other uses that I won't go into but surely a creative person will be glad to look into these possibilities.

Output reports:

Currently CAESAR II has a wide variety of output reports that can be saved as word documents or text files. (See pg 243 of the online users guide) But lets say you want something else… Well without ODBC you could edit the other files cutting, pasting, and deleting information into it manually. This has numerous drawbacks time, and introduction of errors come to mind.

However writing the data you require to an ACCESS™ database and then doing some work in ACCESS allows easy customization. One could combine Node Number, Node name, Restraint Model Coordinate, Restraint Reactions, and Displacements into an ACCESS™ database. Then using the tools in ACCESS™ you can easily create queries reports and control panels, as you need to.

Summary:

I have attempted to simply make people aware of this feature, which can be used quite simply to export data into ACCESS™. Once in ACCESS™ a knowledgeable person can then easily be quite creative in how the incoming data is manipulated and eventually presented.

I did not intend to have this article serve as a how set of instructions in the use of ACCESS™ or EXCEL™. In order to be at the peak of effectiveness in the business of piping flexibility analysis persons involved should be knowledgeable with these wide spread tools and how they may assist in the manipulation of data.

B31.3 – 2004 and CAESAR II

by: Dave Diehl

Introduction

By now, everyone should know that new editions of the ASME B31.3 Process Piping Code are delivered on a two-year cycle with no updates or addenda issued between editions. This new program, started with the 2002 Edition, replaces the previous three-year cycle with annual updates. The content of the 2004 Edition was pretty much defined at the Code Committee meeting in the fall of 2003. The galley proofs for the 2004 Edition were reviewed by all Code Committee members and should be out soon. This new two-year delivery cycle packs numerous of changes into each new Edition. This article reviews the technical changes in B31.3 – 2004 Edition that apply directly to CAESAR II.

The number of Code changes that directly affect stress calculation is impressive. Obviously, the Code Committee now acknowledges that the implicit understanding and experience associated with hand calculations has been totally replaced by engineers' reliance on computer modeling and analysis software. In the past, individuals resolved ambiguities to their own satisfaction and utilized B31.3 paragraph 300(c)(3)[1] as they saw fit. Today, commercial software applications have replaced the step-by-step approach to piping system evaluation. Now it is the authors of commercial computer tools that, at best, resolve these ambiguities or, at worst, mask them further. The background and knowledge of the authors of today's commercial computer tools define the code's gray areas, in effect displacing the judgment of the engineer using the tools[2]. Understanding this, the Code Committee is providing more explicit direction to address these old issues and incorporates the new analysis capabilities readily accessible to all.

Stress Range Reduction Factor, *f*

The limit for the expansion stress range, SA, is defined by the equations (1a) and (1b). All references in this article will be to B31.3 except where noted. Equation (1b) shows $SA = f[1.25(S_c + S_h) - S_L]$.

---

[1] B31.3 – 2002 300(c)(3) "Engineering requirements of this Code, while considered necessary and adequate for safe design, generally employ a simplified approach to the subject. A designer capable of applying a more rigorous analysis shall have the latitude to do so; however, the approach must be documented in the engineering design and its validity accepted by the owner…"

[2] A good example is the calculation of longitudinal (sustained) stress in B31.3. There is no explicit formula provided for this stress and official Code Interpretations regarding the use of stress intensification have varied in the past.

This equation can be broken down into its three components. The $1.25(S_c + S_h)$ term addresses shakedown and low cycle fatigue; S_L reduces this limit to prevent ratcheting failure[3]; and *f*, the stress range reduction factor, incorporates high cycle fatigue considerations.

Now, assuming that the basic allowable stresses, cold and hot, are based on material yield at these temperatures[4] then $S_c = 2/3\ S_Y$ (S_Y is material yield stress) and $S_h = 2/3\ S_Y$ at temperature. The limit for low cycle fatigue range is the sum of yield stress at the "cold" temperature and the yield stress at the "hot" temperature or ($S_Y + S_Y$ at temperature). Exceeding this limit will result in material yield in every cycle and, in short order, failure. In our terms, this converts to $(1.5S_c + 1.5S_h)$. Theory then predicts low cycle fatigue failure when the stress range exceeds $1.5(S_c + S_h)$. The Code permits only $1.25(S_c + S_h)$, thereby providing a factor of safety of 1.50/1.25. Markl's work gives us *f*, the stress range reduction factor, in equation (1c). Equation (1c) reads $f = 6.0(N)^{-0.2} \leq 1.0$ and has a note that also limits *N*, the number of full strain cycles, to no more than 2 million. The values for *f* are conveniently listed in Table 302.3.5 for those not wanting to crank through the equation. We display this table in our help text: less than 7000 cycles (roughly, once a day for 20 years) *f*=1, between 7000 and 14000 cycles *f*=0.9, and so on, to the limit of *f*=0.3 for 2 million cycles.

So what's new in 2004? First, *f* has a new name. In place of stress range *reduction* factor, *f* is now termed the stress range factor. That's because *f* now has an upper limit of 1.2 so there's not much reduction going on if *f* is greater than 1.0. Notice how the theoretical shakedown limit is 1.5 times the $(S_c + S_h)$ sum and the Code limit is 1.25 times the $(S_c + S_h)$ sum? By allowing *f* to increase to 1.2, the resulting allowable stress is back to 1.5 times the sum. Using (1c), you can back-calculate the number of cycles associated with *f*=1.2. That value is 3125 cycles. So, this upper limit of *f*=1.2 is valid for 3125 or fewer cycles. There are limitations to using this higher *f*. These limitations address the reduced fatigue life for high strength steels and restrictions regarding nonferrous materials and materials operating in the creep range. You cannot take *f*>1.0 in setting SA if:

1) the specified minimum tensile strength of the material is greater than 75ksi,

2) S_c & S_h are greater than 20ksi in calculating SA,

3) the material is not ferrous or 4) the material operates in the creep range (T>700F). None of these limitations apply if you keep $f \leq 1.0$.

---

[3] I have also seen references to the Goodman line and its fatigue stress adjustment to explain the S_L reduction.

[4] Paragraph 302.3.2(d) sets S_c & S_h for typical piping materials and 302.3.2(d)(2) defines the 2/3 S_Y limit.

The high cycle limit of 2 million has also been extended. The new *f* goes out to 100 million cycles or what's referred to as "an indefinitely large number of cycles". You may want to call that point the fatigue endurance limit even though the Code Committee chose not to introduce new terms here. The stress range factor at this large number of cycles is 0.15. This matches well with the fatigue limits defined in API Standard 618 (Reciprocating Compressors for General Refinery Services) and ASME OM-3 (Pre-operational and Initial Startup Vibration Testing of Nuclear Power Plant Piping Systems). You may ask, "Why the interest in so many cycles since 100 million over 20 years gives over 13,000 cycles a day?" While you may not see many new land-based process plants on the drawing boards, there's a big business in FPSO (floating production, storage and offloading) systems. On top of the normal thermal cycles, this shipboard piping also experiences the deflections due to the hog and sag of the vessel itself and the inertial loads caused by pitch and roll. One hundred million doesn't sound so big when you're counting waves.

One last thing to be said about this new *f*: in a nod to the ubiquitous computer, the quick reference Table 302.3.5 is now gone. No more stair stepped values for *f*. It's just the equation and its companion plot:

Comparison of B31.3 Cyclic (Reduction) Factors
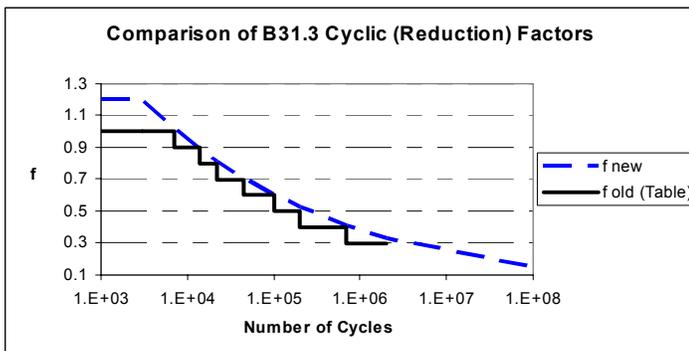
Operating Stress and Alternative Rules

A common question from a user unfamiliar with the Code's stress categories and modes of failure is, **"Where's the allowable stress comparison for the operating load case?"** We usually recite the difference between load-based primary failure limited by yield and displacement-based secondary failure limited by fatigue and how the operating stress is not a measure of either. The 2004 edition now has an allowable limit for operating stress in the new Appendix P. This will make our explanations a little more complicated.

Appendix P is titled *Alternative Rules for Flexibility Analysis* and addresses some of the finer points of analysis in the computer age, namely, the evaluation of stress in piping systems in the presence of nonlinear effects such as support liftoff. CAESAR II's +Y restraint is an easy concept to grasp, but it's very elusive in terms of code-defined stress evaluation. The pipe is free to lift off the support

(Y is vertical in this example), but it is restrained from moving down. Early piping programs (in the early 70's) were totally linear. They had no liftoff capability and the pipe was either free or fixed for all load cases. The linear nature of these tools kept things simple – the expansion stress range was simply the thermal loads plus imposed displacements analyzed alone (T1+D1). The first release of CAESAR II had nonlinear supports and correctly calculated the displacement stress range as the difference between the any two states[5], usually the operating case minus the installed case or L1-L2 in CAESAR II lingo. These nonlinear effects can be troublesome when results from different support configurations need to be combined. For example, how does one handle the situation where the *active* support configuration changes between the installed case and the operating case? CAESAR II will calculate the proper expansion stress range, but the expansion allowable limit (in the Code's equation 1b shown above), with its "-S_L" term, may be based on a different treatment of that +Y support. Some may consider that unconservative. As the active support configuration changes from one state to the next, it is not always clear which state to use in range calculations and their design limits. Appendix P addresses this issue by not mixing states and ranges.

Appendix P is not mandatory and it does not supersede the existing stress evaluation. What it does is replace one stress check with another; one that may uncover overstress conditions that may be lost in the state/range load case shuffle. So what are the alternatives? You EITHER run the current expansion stress range calculations, OR you run the new calculations. Like the base Code, Appendix P checks stress ranges between any two "states" of the piping system (e.g. the range of stress between installation and operation), but it also provides a check for any individual state of stress. Appendix P requires that the largest of these calculations (ranges and states) be less than $1.25f(S_c + S_h)$. Again; all expansion stress ranges and all operating stress states must lie below their respective limits. Note that SL is missing from this allowable limit.

---

[5] The expansion case was always more critical than the sustained case since sustained stresses could always be addressed by wall thickness and support spacing rules. Today, though, the sustained stress seems to be a more interesting issue to some. In fact there is a Code Inquiry asking which support configuration should be used to evaluate sustained stress, and the response is that all possible support configurations should be checked. So, if a pipe rests on a support in the installed state but lifts off the support in any operating state, the sustained stress should be evaluated both with and without that support. CAESAR II, by default, checks only the installed case assuming any liftoff is a secondary effect and is considered in the expansion stress evaluation and any yielding here would be shakedown. You could rerun each plausible sustained stress calculation for each support configuration but a bolder approach would be to turn it all into a single analysis where the sustained stress case is a combination of each operating state minus the thermal range, or (W+T1+P1+D1)-(T1+D1); then (W+T2+P2+D2)-(T2+D2); and so on.

It's now included on the other side of this equation so that the ratchet check (mentioned in the Stress Range Reduction Factor above) is maintained with the operating state stress check.

Appendix P has a few other interesting points. The stresses calculated here explicitly include an axial term. The base Code calculation of expansion stress consisted solely of bending and torsion. Now the range and state stresses follow $\sqrt{\left(\left|S_a\right|+S_b\right)^2+4S_t^{\;2}}$; where S_a, S_b & S_t are intensified axial stress[6], bending stress and torsion stress, respectively. Also, cold spring considerations are tuned up. The base Code allows credit for only 2/3 of a cold spring in calculating initial installation and operating loads on equipment as the exact cut short (or long) cannot be assured. Here, that uncertainty is extended to include an overzealous cut. Appendix P requires examination of 2/3's of design cold spring and 4/3's of design cold spring. This is not a problem in CAESAR II because a user can set the design cold spring in the input and run two cases: W+T+P+.67CS and W+T+P+1.33CS. And, new in Version 5.0, you can assign a temperature-based Young's modulus to the analysis.

Once again, the Code gives a nod to the current capabilities of computer analysis.

Sample Analysis

B31 3 has another new appendix, Appendix S, which illustrates the application of Code rules in stress analysis. Work started a few years ago with a few common problems in stress analysis such as sustained stresses with support lift off and expansion stress ranges with moment reversals (where a hot leg / cold leg switch causes a greater stress range that a general installed-to-operating case). But, there were so many unanswered or unaddressed basic issues in system stress analysis that the Code committee could not resolve these meatier questions until the basic concepts were defined. Some may call it dumbing down or oversimplifying the appendix, but it is more of a solid base from which the Code can build. To review all the considerations addressed by today's pipe flexibility and stress analysis programs is a daunting task and quite an accomplishment for a committee.

The first model presented in Appendix S is a simple code-compliant system; a planar Z bend of constant cross section, no branches, two anchors and a single (linear) Y support. Of course it's a "dull" system, but it sets the groundwork for the future. For one, it sets all "flags" required for the analysis. The configuration settings CAESAR II give an idea of the possible switch settings. Most use default settings but this document cannot rely on vendor defaults.

---

[6] The intensification factor is 1.0 for elbows and i_o from Appendix D for other components.

Items such as pressure stiffening, longitudinal pressure formulation and intensification of moments in sustained stress calculation are all explicitly defined here.

Fig, S301.1
Simple Code Compliant Model
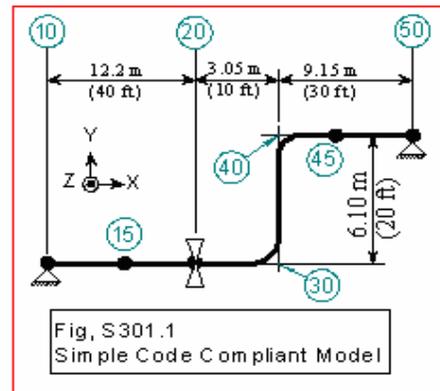
The appendix also runs through a few other calculations or checks before moving on to model input and output and here is where more subtle value exists including a definition of design conditions and analysis conditions, two conditions which may not be identical. Here, too, a second temperature case below the ambient temperature is included. Of course, a wall thickness check for pressure must also be examined before running the analysis. The complete process of setting the minimum wall thickness requirement to checking the actual wall is reviewed.

The model is quite simple and the analysis is almost trivial, but first, a few basic issues had to be settled. For one, a complete stress evaluation would include sustained stresses. So, the code committee took the text in paragraph 302.3.5(c)[7] and converted it to an explicit equation. Questions such as stress intensification and the pressure stress formulation had to be answered. One outcome of this exercise is the proposed B31.3 sustained stress Code Case described below. Including a temperature below ambient made the expansion stress range calculation a little more interesting. CAESAR II users cannot depend on the recommended load cases to produce the same results. CAESAR II recommends (operating case 1 – installed case) and (operating case 2 – installed case) but not (operating case 1 – operating case 2)[8]. It us up to the user to build the overall strain range calculation in CAESAR II. (See M.E.N. May 1993 and July 2003 for details on this procedure.)

---

[7] "The sum of longitudinal stresses in any component in a piping system, due to pressure, weight, and other sustained loadings S_L shall not exceed S_h in (d) below."

[8] Since this system is linear, T1 is identical to (W+T1+P1)-(W+P1) and T2 is identical to (W+T2)-(W). Additionally, since the system is planar, (W+T1+P1)-(W+T2) is identical to the sum of the results of T1 and T2. This will not be true in 3D systems.

The results from three commercial pipe stress programs were used in compiling the results tables. If all the analysis flags are properly set in CAESAR II, your numbers should fall within the stated tolerances. Note that enough tolerance had to be included in the results to allow for the varying round off between metric and U.S. customary input units. The input, configuration, load case and output files for the CAESAR II run of this job can be downloaded from the COADE web site. (Download S_1_SI.zip (for SI units) and S_1_US.zip (for US Imperial units) from the CAESAR II file section.)

Other things

Short term occasional stress allowable in creep range

In accordance with paragraph 302.3.2, the basic allowable stress for some high temperature materials is limited by creep strength ("100% of average stress for a creep rate of 0.01% per 1000 hr") rather than yield strength at temperature ("two-thirds of yield strength at temperature"). Allowable stress controlled by creep places an unnecessary restriction on occasional stresses. Occasional loads are similar to sustained loads in that they are forced-based stresses as opposed to strain-based stresses and they have collapse as their mode of failure. While sustained loads are long term and creep is an issue, occasional loads are temporary so creep is not an issue. Therefore, B31.3 has removed the creep strength restriction from evaluating the sum of stresses due to sustained loads and stresses due to occasional loads. B31.3 has replaced the limit of kSh with up to 90%[9] yield strength at temperature (as listed in the BPV Code, Section II, Part D, Tables Y-1 or Y-3). So, if you have high stress issues with occasional load evaluation, the Code may now accept as safe your higher stresses. This update replaces kS_h but does not alter the value of S_h used in setting kS_h as the allowable stress for sustained plus occasional loads.

Weld Joint Strength Reduction Factors

In elevated temperature considerations, engineers have observed that the strength of weld metal does not match that of the base metal at creep temperatures. This concept is applied as an additional factor in the pressure thickness equations, similar to the weld joint efficiency. Accordingly, all wall thickness (hoop strength) calculations have been modified for welded (not seamless) pipe and welded components. This consideration also plays a role in the sustained stress evaluation of the piping system. The pipe's allowable

stress limit (S_h) must also be multiplied by this weld joint strength joint reduction factor, W. This factor does not affect occasional or expansion stresses[10]. The weld strength reduction factor is the ratio of stress causing failure in the weld material to that of the base metal.

Without your own creep data, set W=1 for temperatures up to 510°C (950°F) and drop linearly to W=0.5 at 815°C (1500°F). You are on your own above 815°C. This concept is applied as an additional factor in the pressure thickness equations, similar to the weld joint efficiency. The application of this factor as a reducer of allowable stress rather than an increaser of calculated stress is awkward. Up until this point, the allowed limit for sustained stress came directly from Appendix A, and an engineer need not locate girth butt welds in the stress analysis. That is no longer the case in elevated temperature piping. CAESAR II does not directly address this weld joint strength reduction.

Code Case defining Sustained Stress

This change is not a 2004 edition per se. Work on this topic grew out of the Appendix S effort. The Code Committee found itself in an odd position when it tried to present an application of commercial stress analysis. Up until this time, the formula for sustained stress was left to the judgment of the individual designer (see paragraph 302.3.5(c)). Any sustained stress used in the Appendix would become the de facto standard simply because it was exercised in the B31.3 document. The sticking point lay in the use of a stress intensification factor in the sustained stress. The two previous Code Interpretations (1-34 from 1981 and 6-03 from 1984) expressed different views[11]. The worked example in Appendix S avoided the issue by keeping the highest sustained stress away from the intensified elbows. Because the issue was already raised, the task group responsible for this work took the steps necessary to generate consensus in the Code Committee. The work was finished too late for the 2004 Edition so, rather than waiting until 2006, this item was instead proposed as a Code Case. Code Cases are not Edition specific so, as passed, the Code Case can go right to the public for

immediate use. This change, written as   , will

be published on the ASME web site when it's official. Terms are the same as defined above, but, here the axial stress is not intensified and the bending moments for all components are adjusted by 0.75i or 1.00, whichever is greater.

---

[9] The variability (72% to 90%) represents the reduction in the long term strength of the material at elevated temperatures. Use 90% times yield for austenitic stainless steels (which do not suffer from this aging) and 72% (actually, $0.8 \cdot 90\%$) for other materials.

[10] It does raise a question though: If $\sigma_{sus} \leq W \cdot S_h$ must be maintained; is the simpler $\sigma_{sus} + \sigma_{occ} \leq k \cdot S_h$ still valid?

Keeping up to date

Strangely, ASME does not notify all purchasers of the previous Code when a new edition is released. When I buy new editions, I ask them to alert us to the new code. However, it's up to you to regularly monitor the new publications list. I recommend that you bookmark the ASME web site at www.asme.org. Remember, B31.3 is on a two year cycle and, while no Addenda are issued, Errata will be posted on the B31.3 web site. You will also find draft documents for comment and Code Interpretations there.

PC Hardware/Software for the Engineering User (Part 35)

File Open Confusion

Recently COADE has received a number of calls and e-mails describing a problem with the "File Open" dialog box. The problem is that the files are no longer in alphabetical order, they appear completely random. This makes finding a specific file difficult, if not impossible in a highly populated directory. What is the solution to this problem?

Actually the solution to this problem is rather simple – it is a "Windows Explorer" setting. To change the setting so that files appear alphabetized in "File Open" dialogs, start Windows Explorer. Pull down the "View" menu, and expand the "Arrange Icons by" option. When this sub-menu expands, click on "Name". Now, the "File Open" dialogs will behave as expected.

You can also access this setting by "right clicking" in the "File Open" dialog box. From the resulting context menu, select the "Arrange Icons by" option.

CAESAR II Notices
TANK Notices
CodeCalc Notices
PVElite Notices

Previous editions of this newsletter contained a list of software errors and omissions for all of our engineering products. In an effort to provide this information in a more timely manner, and to synchronize this information with specific software releases and updates, these lists have been made available on the COADE website, in the download area for each product. These lists will no longer be printed in this newsletter.