Rigorous Analysis, Flexible Design Lead to Creation of Optimal Mudmats

Customized automated simulation tool, integrated with Abaqus FEA, improves efficiency and quality of plate mudmat design for offshore production system

Most people are familiar with images of the upper part of an oil or gas platform visible above the sea water— but there's more complexity at work backstage, beneath the surface. In addition to the pipelines and risers, a mechanical support team— pipeline end termination and/or manifolds, often referred to as PLETs and PLEMs—is needed. And if the sea bottom is too soft to bear the load, which is often the case, the support team may itself need additional support. This is where mudmats come in.

The commonly used plate mudmats, made with carbon steel, consist of a top plate and a number of perpendicular vertical stiffeners that function as load-bearing beams (See Figure 1). But the simplicity of the design belies their importance.

"Mudmats are just as vital as the subsea equipment resting on them," says Dr. Jinming Xu, Engineer at Foster Wheeler Upstream (Houston, Texas). "They must be designed to have sufficient strength, particularly buckling strength, and that requires considerable design analysis."

Since most mudmats are custom-designed for individual load performances, subsea soil properties, and installation requirements, nearly every mudmat requires its own unique stress and buckling analysis. The analysis process used to be largely manual, time-consuming, and unwieldy—until engineers (Xu et al) at Foster Wheeler created their own software, Mudmat*ician*, for plate mudmat design, and integrated it tightly with Abaqus Unified Finite Element Analysis (FEA). The result was an automated process that cut analysis time from weeks to days and significantly reduced the potential for human error in hand calculation.

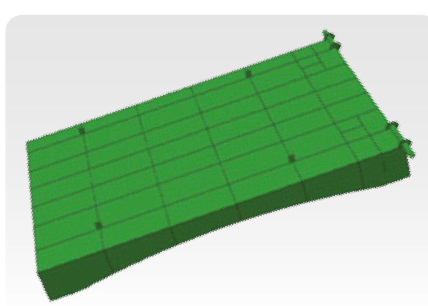


Figure 1. Mudmat*ician*-generated model of mudmat (non-plate accessories added manually), showing top view.

Number crunching during crunch time

Under the old method, designers first obtained panel stresses (checking von Mises stress against the yield stress) directly from an Abaqus FE analysis. Then they had to perform buckling checks (determining panel strength against sudden failure due to buckling) by hand, based on the stresses derived from the analysis at the first step and using industry codes such as API Bulletin 2V to guide the buckling calculation and verification.

To check a single panel, they would extract normal and shear stresses at nine different locations (See Figure 2) based on the panel aspect ratio (length to width). These 27 stresses were then plugged into a long list of complex if-then-else equations that varied with the stress values. In order to properly identify the panels, as well as the stresses at the correct location on each panel, designers had to hand-enter a code for each node and element number.

"The manual intensity of the analysis process made it excessively cumbersome," says Xu, "and like all manual data entry, it could be prone to human error." Also, it was too time-consuming to permit much design modification and iteration, especially given the importance of meeting tight deadlines in the offshore oil and gas industry. "It was clear that we would benefit from shortening the process," Xu adds.

Shorten it they did.

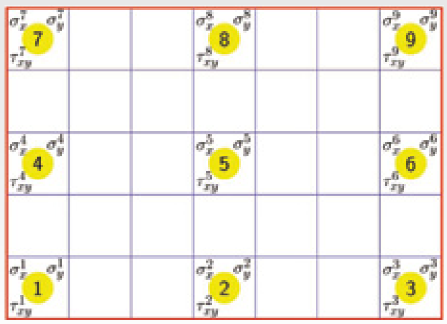
Taking a load off the design cycle

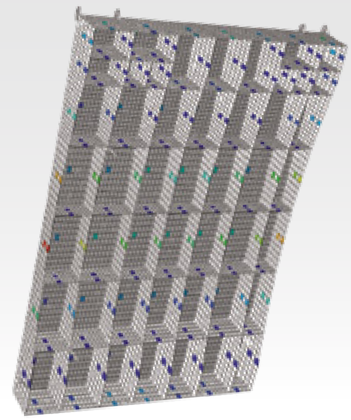
The application programming interfaces (APIs) within Abaqus software helped engineers at Foster Wheeler develop their Mudmat*ician* software. "Abaqus has very good APIs for customized development," Xu says. "It allowed us to write software that let us work with either the geometrical Abaqus model itself or the results in the output database. That's a great advantage, and it enabled us to automate the analyses from finite element model development to post-processing and panel-wise buckling assessment."

In fact, the new automated design cycle for mudmats involves four steps:

1. The designer specifies a series of pre-designed mudmat parameters (size, stiffener layout, and so forth) and creates a Python script design file.

2. The file is then run in Mudmat*ician*, which follows the parameters and generates a meshed model for use in Abaqus/Standard. The model uses spring elements to capture the load-bearing behavior of subsea soil. Model sizes may vary from 10,000 to 100,000 elements.





(Left) Figure 2. A single mudmat panel with 27 normal and shear stresses defined at 9 locations. (Right) Figure 3. Mudmat*ician* output of mudmat buckling checks based on stress analysis results from Abaqus FEA. The utilization factors are shown as a discrete field "B" in contour. Each factor is shown right at the point (element) where it is calculated. The color represents the value of the utilization factor, with least being in blue and greatest being in red. Above, some panels, with utilization factor(s) greater than, are over the buckling limit.

3. The designer adds boundary conditions and loads that are determined by subsea soil properties and by load points for equipment (and, in some cases, by lifting points such as padeyes or trunnions) and a Python script transfers the data into Abaqus. The FEA software then analyzes the mudmat, determining panel stresses.

4. Once the analysis is complete, Mudmat*ician* is employed as a post-processor to extract panel stresses (again, via a Python script) from the Abaqus output database. Mudmat*ician* also performs a buckling check based on the industry code implemented in the software. Although it involves normal and shear stresses at nine different locations per panel, the buckling check is performed only at a few weak points: the center of the panel for the top panels, and the center of the panel and the middle of the free edge for the vertical panels (See Figure 3).

During this process, Mudmat*ician* also reads the design file to identify each of the panels. After each buckling check, results are expressed as utilization factors, the ratios determining whether buckling strength is within acceptable levels. Being less than unity means within the limit, while being greater than unity denotes buckling inadequacy.

When the last step is finished, the designer can evaluate the overall integrity of the mudmat. If the strength seems insufficient or overly conservative, the engineer can return to the first step and quickly analyze a re-designed mudmat. Remodeling is easy and involves changing one or more parameters in the design file to build a new model with Mudmat*ician*, and re-running the FEA and the buckling check. This capability helps designers to easily identify and reduce the likelihood of potential weak points in the mudmat.

Automation = fast, trustworthy results

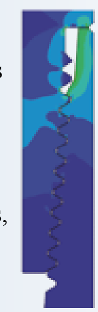
To validate the new technique, the engineers benchmarked Mudmat*ician* analyses against existing manually designed mudmats and compared the manual and software results. The correlation was good, and the first mudmat designed entirely with Mudmat*ician* has now been completed. "The amount of man-hours spent on the design using Mudmat*ician* was dramatically reduced," says Xu. "What once took weeks or even months now only takes days. That gives us the freedom to re-design if necessary, while meeting tight deadlines."

In one case, the new automated analysis path enabled Foster Wheeler Upstream to offer emergency services to a client who requested an extremely rapid mudmat design for an urgent project. "We wouldn't have been able to help them so quickly without Mudmat*ician* and Abaqus," Xu says.

"Together, Mudmat*ician* and Abaqus have streamlined our analyses and shortened our design process," Xu says. "With improved engineering efficiency and design modification, it is now much easier to optimize mudmats. This allows us to deliver the highest-quality mudmat designs to our clients, quickly."

Abaqus Extension for Threaded Connections

Customized extension for Abaqus/CAE accelerates modeling and analysis of threaded connections



The performance of threaded connections is critical in several industries including the Oil & Gas industry, where they must operate reliably under high pressure and temperature conditions. FEA can be used to improve the reliability of threaded connections, but simulation set-up can be complicated, requiring accurate representation of the thread, box and pipe geometries, as well as the material properties. Often, analyses are not limited to a single connection, but need to be carried out for an entire family of connections.

The Threaded Connection Modeler (TCM), one of many Abaqus Extensions offered by SIMULIA, was created by our Southern region office in the US to provide a tool for quickly building entire families of threaded connection models and performing the make-up and loading steps.

The TCM plug-in is implemented using a Model-Tree approach. As each step in the process of building a threaded connection is completed, icons throughout the model tree are modified to provide immediate feedback on the user's progress. As an integrated Abaqus plug-in, all of the features of Abaqus/CAE can be used to enhance the model.

Additional Extensions for Abaqus

The robust architecture of Abaqus FEA provides users with the ability to create custom interfaces and automate analysis processes. In addition to the Abaqus extension for threaded connections, our regional offices have developed and supported a variety of additional extensions for specific industry applications, including the Abaqus Welding Interface, Adjustable Rigid Torus (ART), and Bolt Studio.

For More Information
www.fwc.com
www.simulia.com/products/extensions