PUTTING HIS BEST FOOT FORWARD

Ritsumeikan University researcher employs Abaqus for a realistic model of flatfoot deformity to help surgeons plan treatment

University researcher Dr. Wang Zhongkui has only been using SIMULIA's software for a little more than a year. However, as a Ph.D. candidate from China who came to Japan's Ritsumeikan University to study robotics, he had been working with other forms of FEA for a long time—very long, in his opinion.

"I used to make my models by myself using my own code, but that would take weeks to prepare the mesh, make the program, and finally finish a simulation. Plus, the post-processing and visualization were not very graphical or pretty," Dr. Wang says.

"When I started working on the challenge of surgical planning for flatfoot, I did some background research and discovered that most people who were modeling biology cases were using Abaqus," he says. "Human tissue modeling is one of the toughest problems because many of the physical properties are nonlinear. That is, of course, a strength of Abaqus—as well as quality graphics and very nice pre- and post-processing for deformation, stress and so on."

Dr. Wang's flatfoot project came into being when doctors at nearby Kyoto Prefectural University of Medicine contacted Prof. Hirai Shinichi, who had been Dr. Wang's supervisor since he came to Japan, asking for help with computer models that could be used for surgical planning to correct severe cases of this common human ailment (see sidebar). Japan is well known for its robotics expertise—and simulations of robots, including detailed models of feet, were a strength at Ritsumeikan. Dr. Wang had finished his Ph.D. by then, and had already started to work on life-science-related simulations for human applications, so the doctors' request found him willing to take on the challenge.

"Deep in my heart, I've always wanted to help people," he says. "By working on medical-related problems with simulation I feel we can make some contribution."

TAKING STEPS TOWARDS A MORE REALISTIC SOLUTION

Dr. Wang had developed his previous FEA models to simulate balanced standing, but contact and friction among neighboring bones were not included, and the gaps among bones were connected by soft tissue, resulting in relatively stiff joints that affected model performance. Once the request came in from Kyoto, he knew he needed to step-up his simulation capabilities.

When he switched to Abaqus, Dr. Wang saw remarkable improvements in time as well as quality. "I'm really happy now," he says. "I don't need to spend a lot of time preparing the mesh and working out the equations," he says. "I feel like getting from setup to results is now about ten times faster."

Dr. Wang built his Abaqus model of a single right foot by using geometries of bones and tissue generated from CT images of a 168 cm tall, 62 kg male volunteer (see Figure 1). Ligaments, plantar fascia, and tendons were modeled as either 3D solid parts or tension-only wire parts. Because even with CT images it can be difficult to discriminate these tissues, he referenced the positions of these various components from both anatomy books and instructions from foot surgeons.

All of the materials in the foot model were considered isotropic and linearly elastic except for the encapsulated tissue, which was modeled as both linear and nonlinear elastic materials for comparison. The tibia and fibula bones of the lower leg were fixed in space, and a ground plate was loaded and pushed upward with half of the body weight (since each foot carries only that amount) to simulate a balanced standing position. A ground-reaction force and an Achilles-tendon force were also included in the calculations.

Dr. Wang ran his simulations with different material properties of tissue and different boundary conditions for the ground plate, then compared the resulting stress maps against experimental measurements where the volunteer stood on a pressure platform system (see Figure 2).

LESSONS LEARNED ALONG THE WAY
One thing Dr. Wang found was that modeling ligaments and plantar fascia with wire parts (truss elements) produced almost the same performance as 3D solid parts. "So in future work we'll model with wire parts because they are easier to construct," he says.

He also learned that the material properties of the tissue, and the boundary conditions of the ground plate, had distinct effects on results, so he subsequently fine-tuned the boundary conditions to get the best model performance. He further determined that nonlinear simulation of encapsulated tissue produced more accurate results than linear.

Finally, he realized that the flat plantar surface he had been basing his models on was less than ideal for consistency with experimental results (see Figure 3). "We think a relatively curved surface will improve our predictions so we'll investigate that in our future work," he says.

Next steps for Dr. Wang include validating his flatfoot model and developing simulations of different types of surgery. His team is also going to create 3D-printed, multi-material prototypes from their FEA models. "Abaqus should be able to help us optimize the STL files we use for the 3D printing," he says.

"My hope is that all our work will help doctors better visualize foot-related issues and develop optimal surgery plans that are tailor-made for individual patients," he says.

GETTING A LEG UP ON THE COMPETITION
So, with everything going on in his laboratory, why did Zhongkui take the time to enter an SCC academic poster contest? "When I saw the Call for Posters [which SIMULIA sends out to all academic users every year] I decided to participate in the poster session because I wanted more chances to communicate with peer engineers and learn from them," he says. Although he also gave a formal presentation at the event, he liked the opportunity the poster session provided for discussing the details of his research in a more casual way.
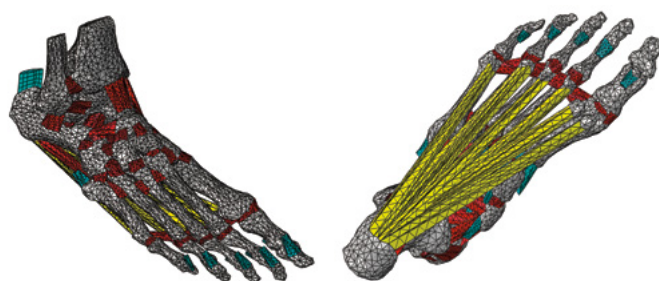

Figure 1. (Left) tarsal and (right) plantar views of the generated 3D solid ligaments (red), plantar fascia (yellow), and tendons (cyan) of Dr. Wang's Abaqus models of a human foot.
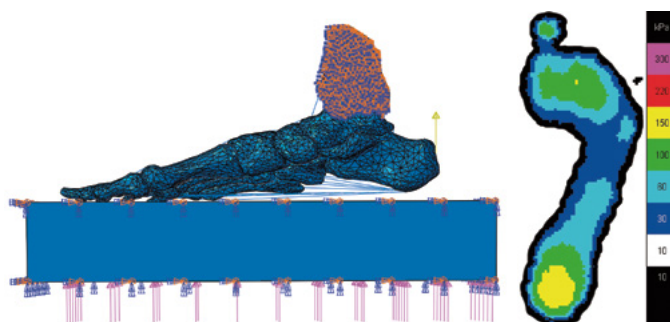

Figure 2. (Left) Loading and boundary conditions defined in the model and (right) measured plantar stress distribution.
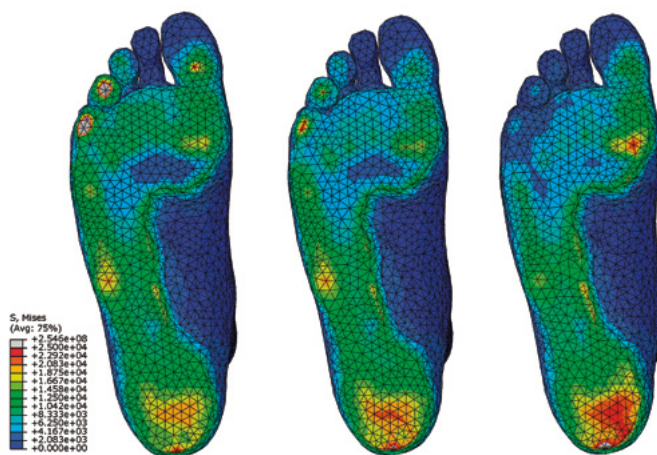

Figure 3. Simulated plantar stress with different boundary conditions. The model at far right, which employed nonlinear hyperelastic tissue, showed the closet agreement to experimental values.