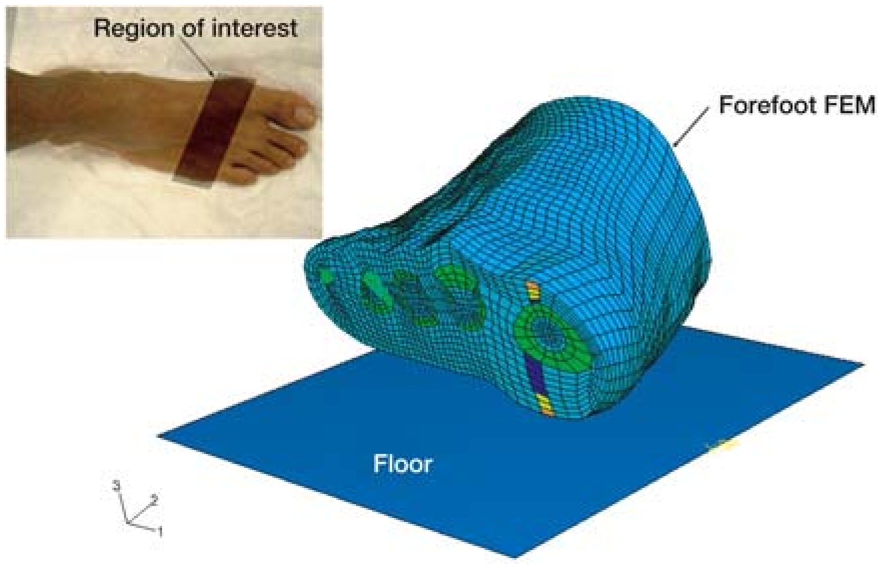
Finite Element Modeling Used to Study Stress Distribution on the Foot

A method to study the stress distribution inside the forefoot during walking was developed at the Cleveland Clinic Foundation by a researcher from the NASA Glenn Research Center. In this method, a semiautomated process was outlined to create a three-dimensional, patient-specific, finite element model (FEM) of the forefoot using magnetic resonance images (MRI). The images were processed in Matlab using the k-nearest neighbor (k-NN) classification algorithm and Sobel edge detection to separate the different tissue types: bone, skin, fat, and muscle. This information was used to create curves and surfaces that were exported to an FEM preprocessor known as Truegrid. In Truegrid, eight-noded or brick elements were created by using surface mapping. The FEM was processed and postprocessed in Abaqus. Material properties of the models were obtained from past experiments such as fat pad confined compression, skin axial and biaxial tests, muscle in vivo compressive tests, and reference literature (bone properties). Nonlinear (hyperelastic) material models were used for the skin (epidermis and dermis), fat, and muscles; and a linear elastic model was used for the bones.

Muscle activation during walking yielded uncertainties in the muscle material model since contracted muscles are stiffer than relaxed muscles. These uncertainties were resolved by performing a sensitivity analysis of the muscle material properties. The original properties were multiplied by arbitrary factors of 2, 3, 0.5, and 0.33. The strain and stress distributions, as well as the locations of peak values, were similar in all cases. The peak contact pressure P obtained for each case varied with respect to the applied factor f as follows:

$$P = 0.2563f + 0.3331 \quad (R^2 = 0.9885)$$

where R^2 is the coefficient of determination.

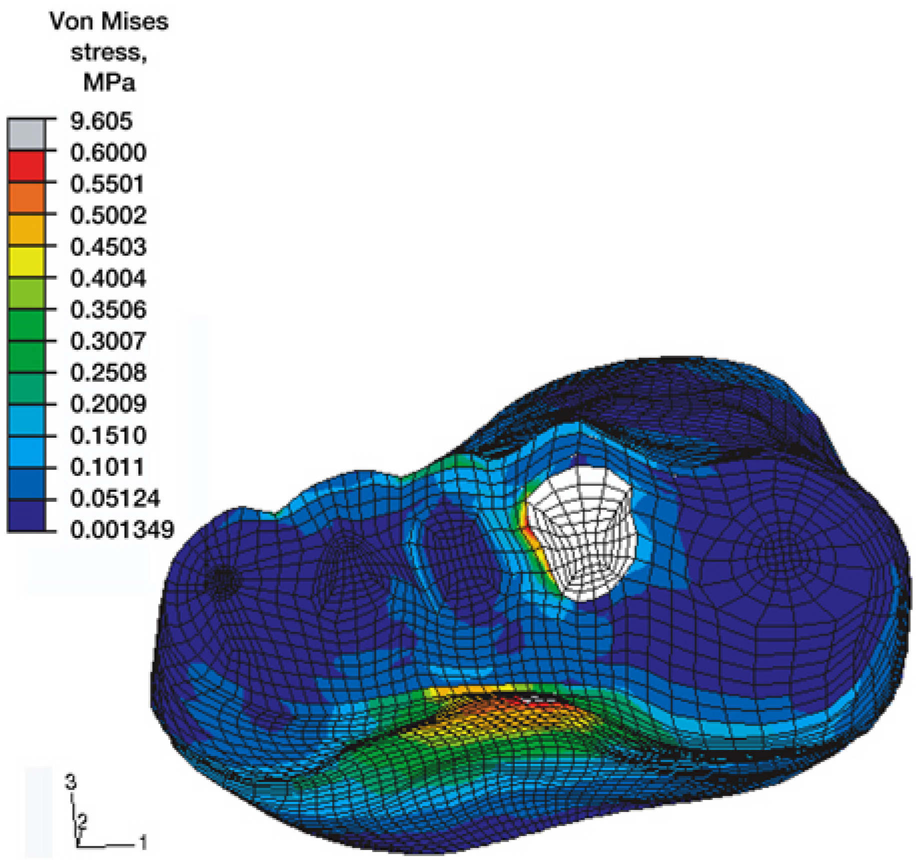


FEM configuration of the forefoot. Inset shows the region of interest in the forefoot for which the model was created.

Long description. The region of interest is bounded by part of the arch of the foot and the ball of the foot (joint of metatarsal heads and phalanges). This model does not include the toes. The model is shown as assembled for the surface contact calculations. The plane, marked as the floor, is a rigid surface that is moved towards the forefoot model during the finite element model calculations. The finite element model of the forefoot includes material properties for the bones, skin (epidermis and dermis), fat, and muscles. The skin was represented with the first layer of elements around the forefoot and is marked by an orange element. The second and third layers of elements were fat, marked by a yellow element. The outer boundaries of the five metatarsal bones are shown in green. The rest of the elements are considered part of the muscles (blue).

The forefoot FEM was validated using a contact model in which a rigid surface (floor) was pressed against the bottom surface of the foot, as shown in the preceding figure. This simulation captured the stance phase of the gait cycle. The contact pressure obtained from this model was compared with experimental pressure data. The pressure profiles for both cases were very similar even though the magnitudes were different (experimental data--0.295 MPa; FEM--0.624 MPa). The location of the peak values was similar in both cases. This difference could be due primarily to inaccuracies in modeling the muscle activation levels. Also, other inaccuracies may exist since the material models were obtained using tests with loads different than the loads during walking (experimental data--uniaxial tension; simulation--three-dimensional tension, compression, and shear)

After validation, peak loads were applied to study the interaction between tissue layers. The following figure shows the results of this simulation. Although all these different tasks were done using a nondiabetic foot model, the same could be repeated for a diabetic foot model if the appropriate material properties were adjusted accordingly. After repeating similar tasks for a diabetic case and comparing them with a nondiabetic case, we found that the internal Von Mises stresses were much higher in the diabetic case (4 MPa vs. 0.6 MPa) even though peak pressure loads were 30-percent higher in the diabetic case (0.9 MPa vs. 0.7 MPa).



Von Mises stress distribution inside the forefoot region. Average criterion, 75 percent.
Long description. This is a Von Mises stress plot (nondiabetic case) of the forefoot finite element model after applying pressure loads. The figure shows a front view, which is a cross section of the forefoot around the metatarsal head joint. The maximum Von Mises stress was approximately 0.6 megapascals and was located on the region between the second and third metatarsals, while the stress on the rest of the forefoot was less than 0.15 megapascals. These results are consistent with the pressure data obtained during the pressure load measurements. Also shown is the stress distribution inside the foot, between the different tissue layers (skin, fat, muscle, and bones). The largest difference in stress across elements can be seen at the skin-fat interface below the third metatarsal head (approximately 0.5 megapascals on the skin side and 0.1 megapascals on the fat side).

Bibliography

Morales, Nelson: Stress Distribution Study on the Foot Using Finite Element Modelling. Master of Science Thesis, Case Western Reserve University, 2003.

Find out more about this research:
John Glenn Biomedical Engineering Consortium at
http://microgravity.grc.nasa.gov/grcbio/bec.html
Lerner Research Institute, Department of Biomedical Engineering at
http://www.lerner.ccf.org/bme/
Lerner, Musculoskeletal Biomechanics projects at
http://www.lerner.ccf.org/bme/biomechanics/

Glenn contact: Nelson Morales, 216-977-7050, Nelson.Morales-1@nasa.gov
Authors: Nelson Morales, Brian Davis, and Azita Tajaddini
Headquarters program office: OBPR
Programs/Projects: JDRF, John Glenn Biomedical Engineering Consortium, BEGIN