ANSYS - Principal Stresses in 2D

How are principal stresses reported for 2D plane stress cases in ANSYS Mechanical? Why is one principal stress not zero everywhere?

ANSYS reports three principal stresses in 2D with one of these always being zero at ANY PARTICULAR LOCATION. Which of the three is zero can change depending on the state of the stress at a particular location. the The three principal stress values are ordered from highest to lowest taking into account the sign. So one could have three situations:

1. Both non-zero principal stresses are tensile

sigma1 > 0, sigma2 > 0, sigma3 = 0

2. One principal stress is tensile and other is compressive

sigma1 > 0, sigma2 = 0, sigma 3 < 0

3. Both non-zero principal stresses are compressive:

sigma1 = 0, sigma2 < 0, sigma3 < 0

Thus, the zero principal stress can be classified as max, middle or min principal stress depending on the state of the stress at a particular location. So you really need three principal stress plots even in 2D.