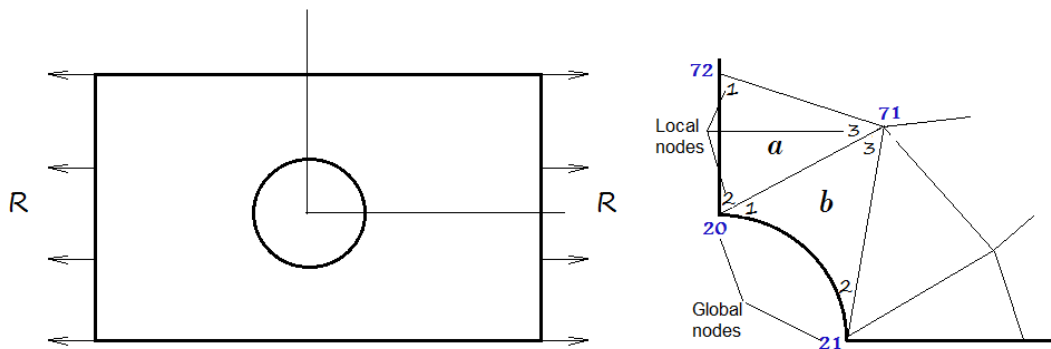


First the FEA computation process in ANSYS:

We have two types of nodes involved in an FEA solution. Example shown is a plate with circular hole and horizontal tensile load applied at the two ends. Due to symmetry only a quadrant is simulated using (say) ANSYS. We have shown **triangular elements**. There are two types of nodes:

Local nodes (shown inside the elements)

Global nodes (shown outside in blue color)



First stiffness matrices are formed at element level. For example the **element stiffness matrices** for the elements 'a' and 'b' can be written in terms of **local load vectors** $[r_1 \ r_2 \ r_3]_a^T$, $[r_1 \ r_2 \ r_3]_b^T$, local stiffness values k_{ij} , and **local displacement vectors** $[d_1 \ d_2 \ d_3]_a^T$, $[d_1 \ d_2 \ d_3]_b^T$. Here 1, 2, 3 denote the **local nodes** for the elements 'a' and 'b'. Then by applying the fundamental concepts of **Equilibrium** and **Compatibility** we assemble the **element stiffness matrices** for all elements over the domain to obtain the **global stiffness matrix** system as shown in the figure attached. Here $[D_1 \ D_2 \ \dots \ D_N]^T$ and $[R_1 \ R_2 \ \dots \ R_N]^T$ are the **global displacement** and **load vectors** in terms of **global nodes**.

Element Stiffness Matrices

$$\begin{aligned} \text{Element 'a': } \begin{bmatrix} r_1 \\ r_2 \\ r_3 \end{bmatrix}_a &= \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix}_a \begin{bmatrix} d_1 \\ d_2 \\ d_3 \end{bmatrix}_a \\ \text{Element 'b': } \begin{bmatrix} r_1 \\ r_2 \\ r_3 \end{bmatrix}_b &= \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix}_b \begin{bmatrix} d_1 \\ d_2 \\ d_3 \end{bmatrix}_b \end{aligned} \quad \text{Assembly} \Rightarrow$$

Global Stiffness Matrix

$$\begin{bmatrix} R_1 \\ R_2 \\ \vdots \\ R_{20} \\ R_{21} \\ \vdots \\ R_{71} \\ R_{72} \\ \vdots \\ R_N \end{bmatrix} = \begin{bmatrix} K_{1,1} & K_{1,2} & K_{1,20} & K_{1,21} & \dots & K_{1,71} & K_{1,72} & \dots & K_{1,N} \\ K_{2,1} & K_{2,2} & K_{2,20} & K_{2,21} & \dots & K_{2,71} & K_{2,72} & \dots & K_{2,N} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ K_{20,1} & K_{20,2} & K_{20,20} & K_{20,21} & \dots & K_{20,71} & K_{20,72} & \dots & K_{20,N} \\ K_{21,1} & K_{21,2} & K_{21,20} & K_{21,21} & \dots & K_{21,71} & K_{21,72} & \dots & K_{21,N} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ K_{71,1} & K_{71,2} & K_{71,20} & K_{71,21} & \dots & K_{71,71} & K_{71,72} & \dots & K_{71,N} \\ K_{72,1} & K_{72,2} & K_{72,20} & K_{72,21} & \dots & K_{72,71} & K_{72,72} & \dots & K_{72,N} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ K_{N,1} & K_{N,2} & K_{N,20} & K_{N,21} & \dots & K_{N,71} & K_{N,72} & \dots & K_{N,N} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \\ \vdots \\ D_{20} \\ D_{21} \\ \vdots \\ D_{71} \\ D_{72} \\ \vdots \\ D_N \end{bmatrix}$$

The overall problem is solved in terms of global nodes and solution is obtained as $[D] = [K]^{-1}[R]$. Once **displacement vector** is computed we can compute **strain components** using the **spatial derivatives of the displacement**. Once strain at each node (global as well as local nodes) is known we can compute **stress σ** at each node.

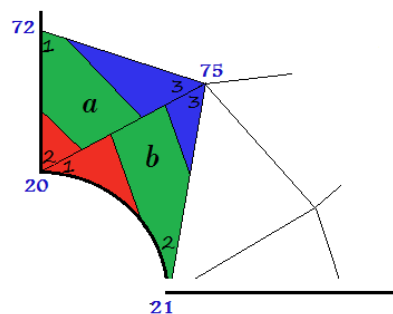
ANSYS displays stresses in two ways: Element stress solution and Nodal stress solution. **Element stresses solutions:** Here the element stresses are displayed within individual elements. They are non-averaged stresses. The stress distribution is unique to an individual element. Understanding **local node** distribution is important for this.

Nodal stress solutions: Here stresses are given to you in averaged form at each global node. **The stress value at a global node is average of all the local-node-stress-values of all the elements sharing that global node** as explained in the table in the attached figure.

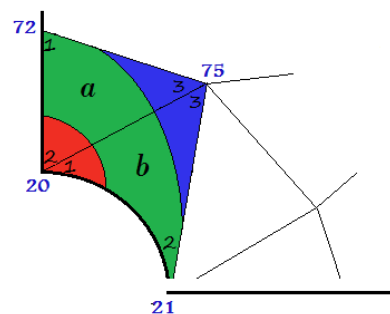
The **discontinuous type of stress distribution in element solution** only indicates that **you need mesh refinement**. As you go on refining the mesh, the element solution looks more and more like the nodal solution.

Values of stress at local nodes of elements 'a' and 'b'

Element	local node	stress	Color bar
a	1	70	
	2	120	
	3	10	
b	1	110	
	2	90	
	3	30	



Element Stress Solution, σ_x



Nodal Stress Solution, σ_x