

# MATLAB - redAnTS 1

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Comments](#)

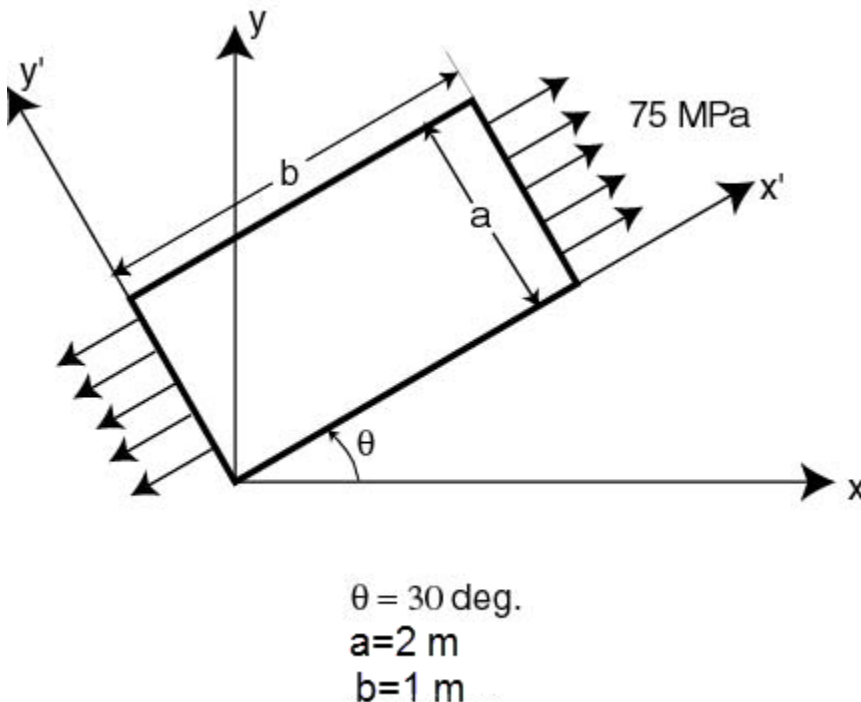
## Introductory redAnTS Tutorial

This is an introductory tutorial on the *redAnTS* finite-element analysis (FEA) toolbox that runs within MATLAB. After the toolbox is installed, it can be called from MATLAB by typing *redAnTS* at the MATLAB prompt. This tutorial takes you through the steps for solving a simple problem with a known solution from elementary theory. This is handy since we can validate the FEA solution by comparing it with the corresponding theoretical result. After you have done a number of FEA solutions for basic problems and validated the results against theory or handbook values, you'll be off running and ready to tackle more complex problems that don't readily yield to paper-and-pencil calculations. (But don't throw the pencil out since it is absolutely essential to verify the computational results with back-of-the-envelope calculations. Consider this as advice from somebody who has been bitten many times in uncomfortable parts of the anatomy by easily trusting results that the computer spits out.)

Basic familiarity with MATLAB is assumed; so keep your MATLAB wits around as you indulge in this tutorial. Happy computing!

### Problem Specification

The humble rectangular block shown below is under uniaxial tension. Note the dimensions of the block. The applied normal traction on faces with normals parallel to  $x'$  is 75 MPa. The Young's modulus of the material is 200 GPa and the Poisson ratio is 0.3. We'll assume that plane stress conditions apply.



We'll employ friendly *redAnTS* to solve this problem using FEA. We'll plot various components of stress and strain for the block and compare them with theory.

### Solution Steps

The *FEA* solution is divided into the following steps:

- Step 1: Pre-Analysis & Start-Up

- Step 2: Geometry
- Step 3: Mesh
- Step 4: Setup (Physics)
- Step 5: Solution
- Step 6: Results
- Step 7: Verification and Validation

Let's dive into [step 1](#) without further ado.

**[Go to Step 1: Pre-Analysis & Start-Up](#)**

[Go to all MATLAB Learning Modules](#)