This folder contains functions to compute the joint response:

Bisection.m

CalcBreakpoints.m

CalcForceStatusDir.m

Intersect.m

Line.m

RichardEquation.m

SingleBoltAngleJoint.m

The folder also has the hysteresis response of the joint from Abaqus Simulation:

Load History: Upeak=[0 0.04 -0.04 0.06 -0.06 0.08 -0.08 0.13 -0.13 0.18 -0.18 0.3 -0.18 0.4 -0.18 ];

H1\_4.mat (L 2\*2\*1/8 )

Load History: Upeak=[0 0.04 -0.04 0.06 -0.06 0.08 -0.08 0.13 -0.13 0.18 -0.18 0.3 -0.3 0.5 -0.3 ];

H2\_4.mat (L 2\*2\*3/16 )

Load History: Upeak=[0 0.04 -0.04 0.06 -0.06 0.08 -0.08 0.13 -0.13 0.18 -0.18 0.23 -0.23 0.28 -0.28 ];

H3\_4.mat (L 2\*2\*1/4 )

Load History: Upeak=[0 0.04 -0.04 0.06 -0.06 0.08 -0.08 0.13 -0.13 0.18 -0.18 0.23 -0.23 0.28 0 ];

H4\_4.mat (L 2\*2\*5/16 )

H25\_1.mat (L 2 ½ \*2 ½ \* 3/16)

H25\_2.mat (L 2 ½ \*2 ½ \* 1/4)

H25\_3.mat (L 2 ½ \*2 ½ \* 5/16)

H33\_1.mat (L 3 \*3 \* 3/16)

H33\_2.mat (L 3 \*3 \* 1/4)

H33\_3.mat (L 3 \*3 \* 5/16)

Main file is:

Test.m

To run the program:

1. Users need to open Test.m and update the Input Variables accordingly in the input variables. From line 6 to line 18.

2. Urecord is the history of the displacement which is depend on Upeak which can be defined by users.

3. Frecord is the computed response of the joint.

4. To compare the zero-length-element response with Abaqus results. Users need to uncomment line 68 70 and 71. On line 70 user need to specify the joint configuration supposed to be plotted. For example, if user want the hysteresis of L3\*3\*3/16 joint, Line 70 will be :

load('H33\_1.mat','U','F');

And the input variables from line 6 to line 18 need to be changed to width\_brace=3 and thickness\_brace=3/16.