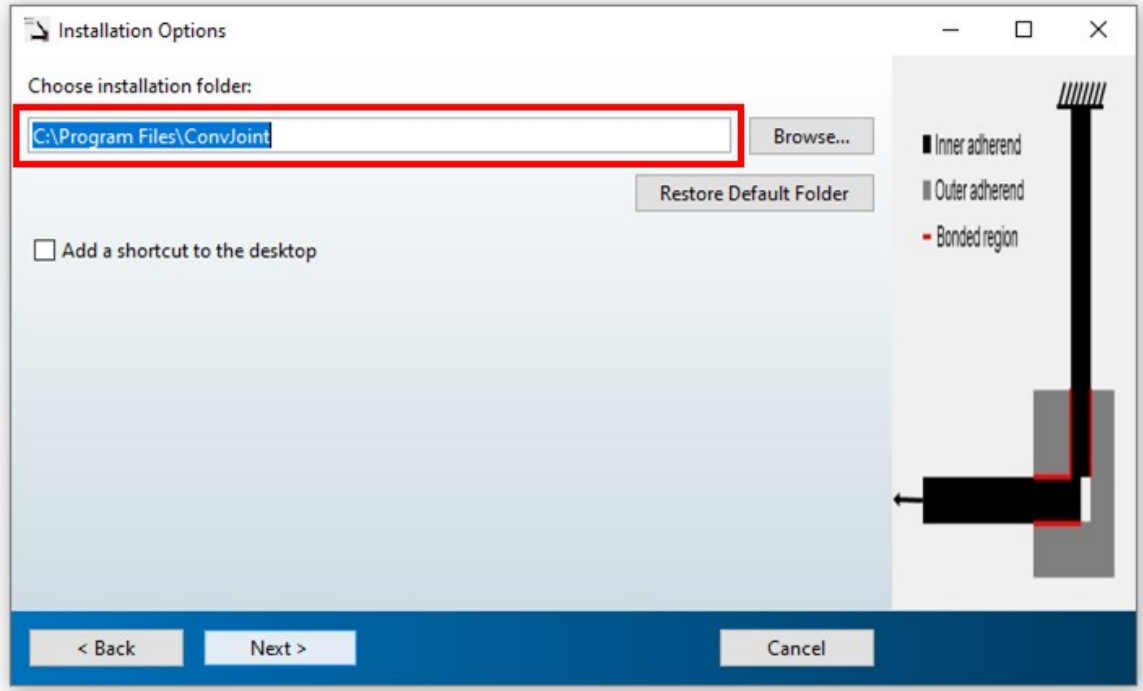
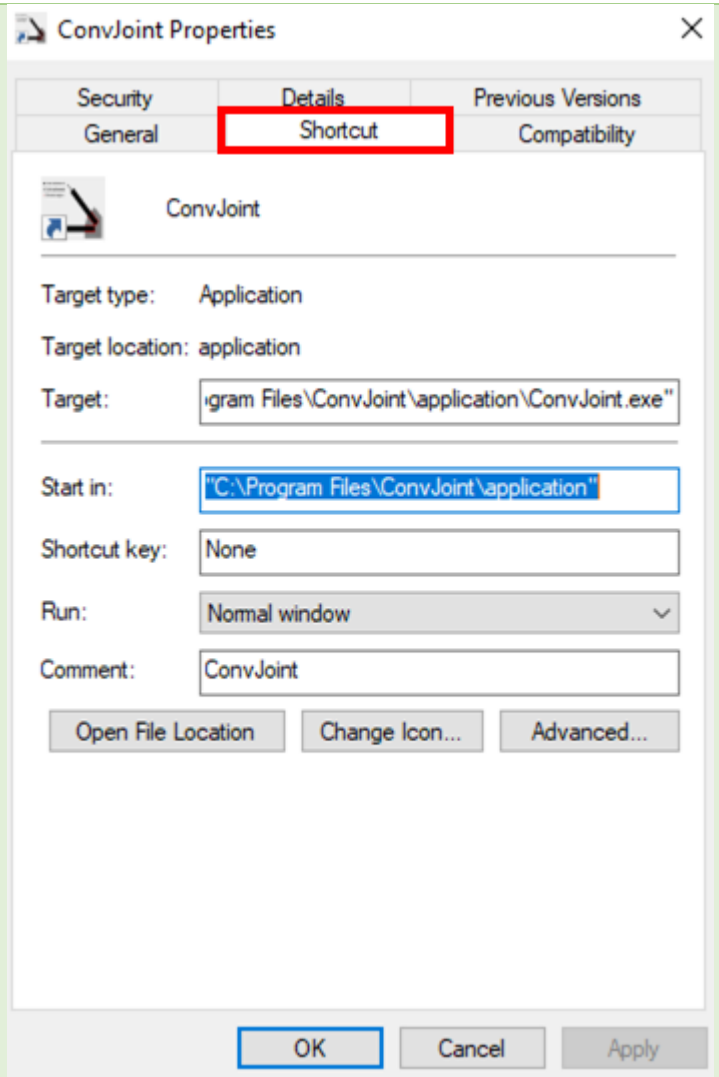
# Conv Joint User Guide

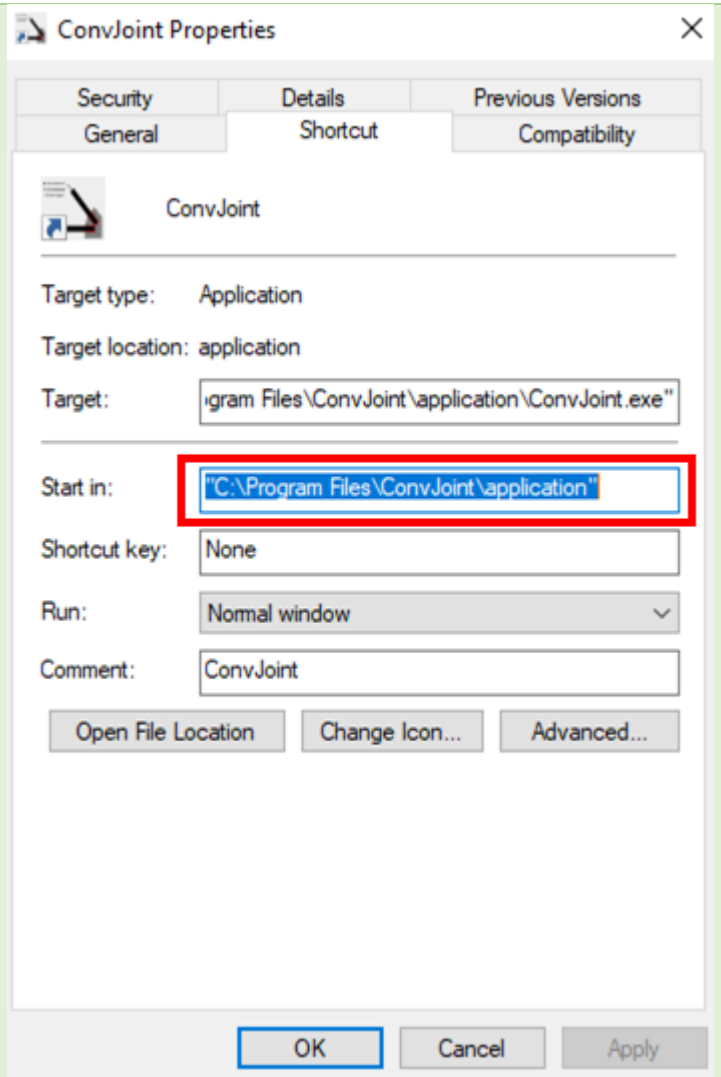
## 1. Installation

| | |
|---|---|
| **Step 1.** | Run the '***ConvJoint Installer.exe***' file |
| **Step 2.** | Make a note on the selected installation folder |
| **Step 3.** | Once installation is complete,<br>If you chose to create a shortcut to the desktop<br>Right-click on the shortcut and click on '***Properties'*** |



| | | |
|---|---|---|
| | If no shortcut was created skip to Step 6 | |
| | | |
| **Step 4.** | In the Properties dialogue box, select '***Shortcut'*** |  |

**Step 5.** Modify the content of '***Start in'*** to match the installation folder selected in Step 2, including '***application***' as shown in the image



| | |
|---|---|
| **Step 6.** | Locate the application in the start menu, right-click on it and select '***Open file location'*** |
| | |
| **Step 7.** | Right-click on the application '***ConvJoint***' in the file location and click on '***Properties'***<br>Perform Step 4 and Step 5 |

## 2. User Guide

This program determines how effective the joint would be when subjected to a loading condition.
The determination is done using a trained convolutional neural network (CNN) which outputs a value ranging from 1 to 10, where 1 indicates poor effectiveness and 10 good effectiveness to the applied load.
If available, the commercial product **ABAQUS** can be used to verify the data provided by the CNN. Its output value would be normalised to fit the value range produced by the CNN.
A database has been created to allow for usage of the program without an ABAQUS license.
However, an ABAQUS license would be required to generate the designs/models not included in the database.

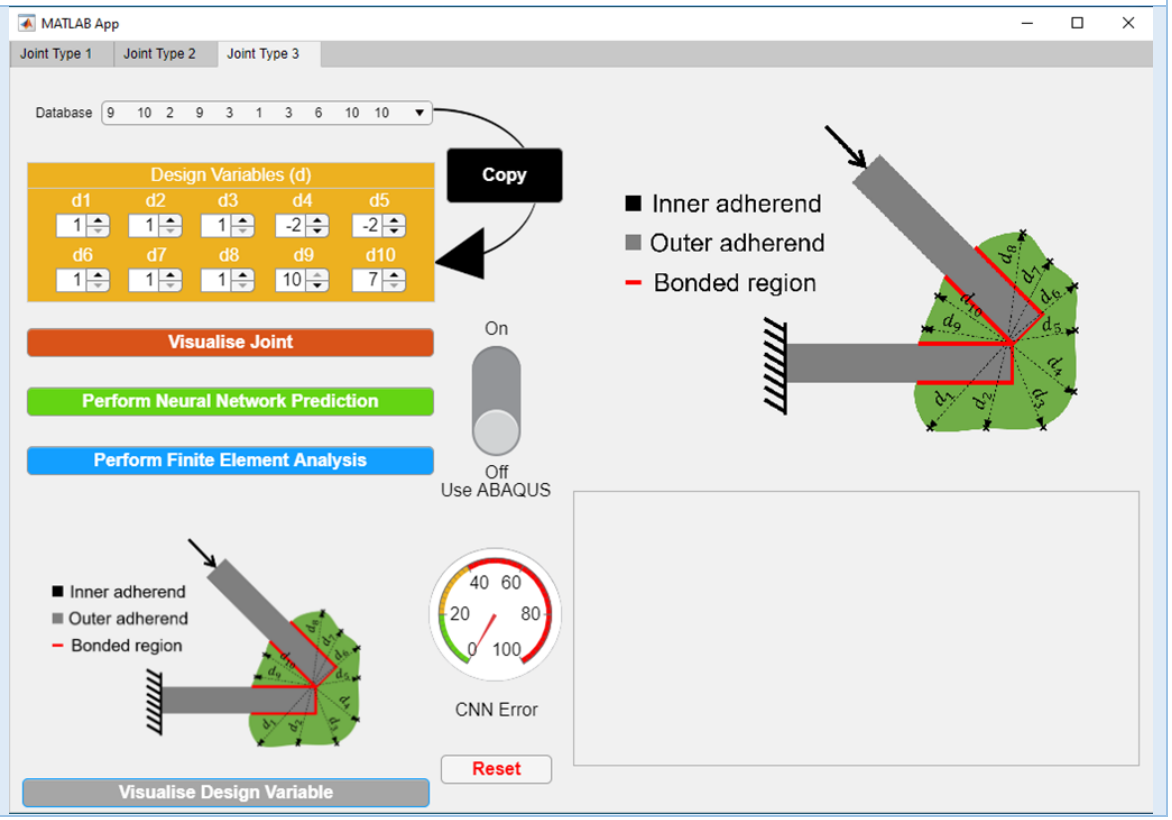
**Step 1.**

The '**Reset'** button can be used to reset the application.
This button can be used if the application is unresponsive.

The '**Visualise Design Variable**' button shows a diagrammatic explanation of how the design variable affects the joint shape

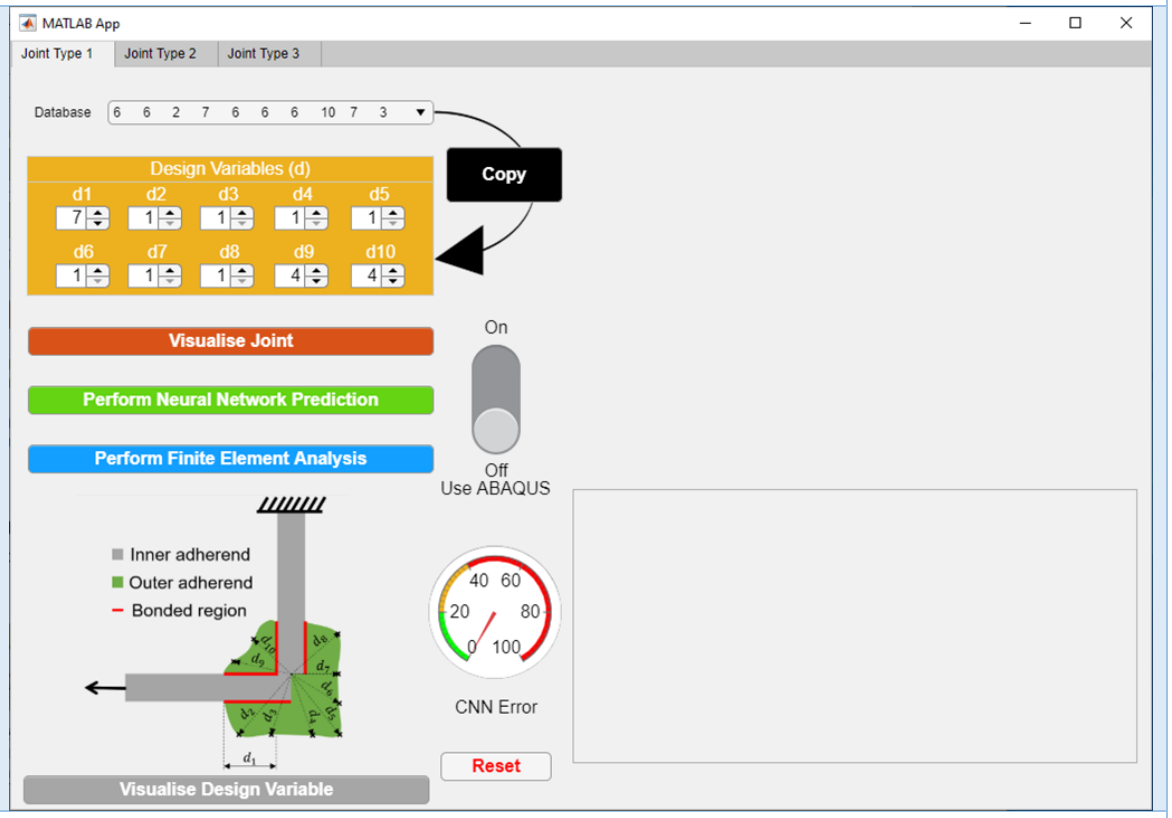
The '**Perform Neural Network Prediction**' button shows performs the prediction of the normalised specific reaction force using the CNN and displays the results

The 'Perform Finite Element Analysis' button predicts the normalised specific reaction force value using the CNN and performs the finite element analysis for verification and comparison.



**Step 2.** Open the Application

*The first time the application is opened, it might take a couple of minutes to open.*

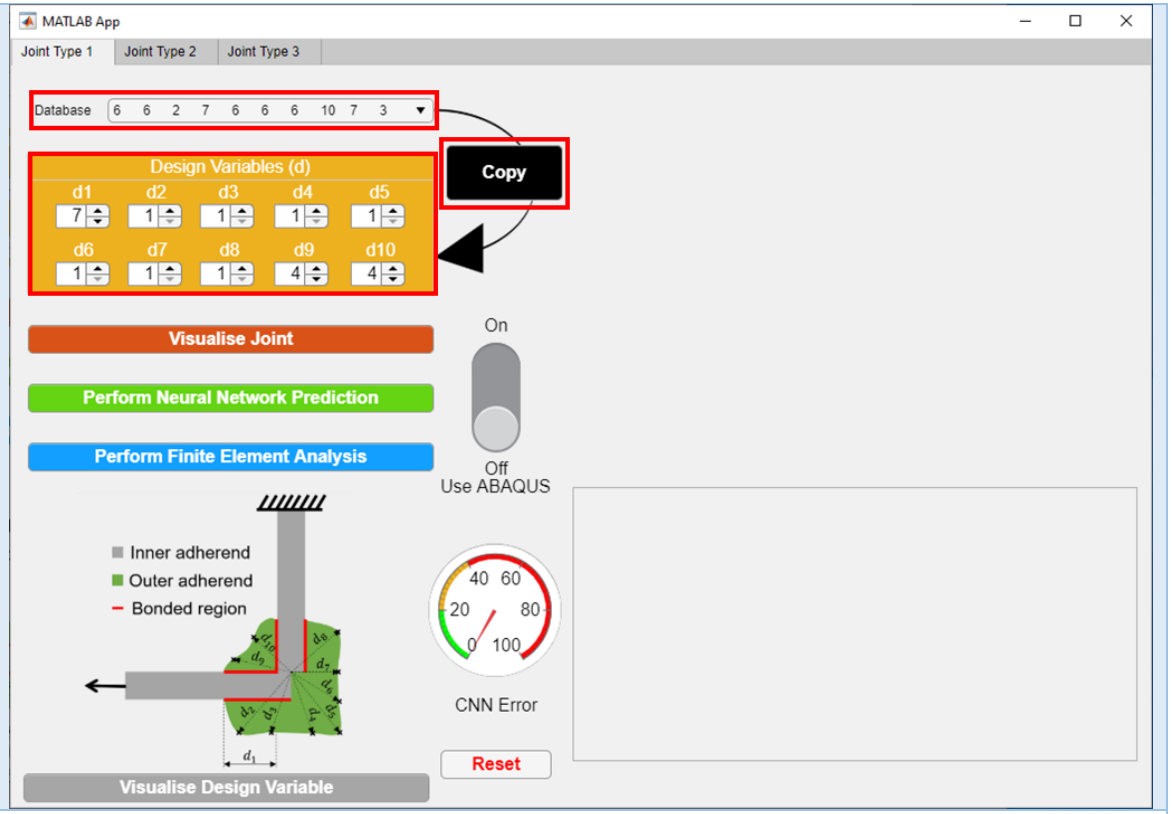


**Step 3.**

There is a database available for each Joint type.

The user can select the design variable from the database and press the **copy** button to move it to the current design variables.

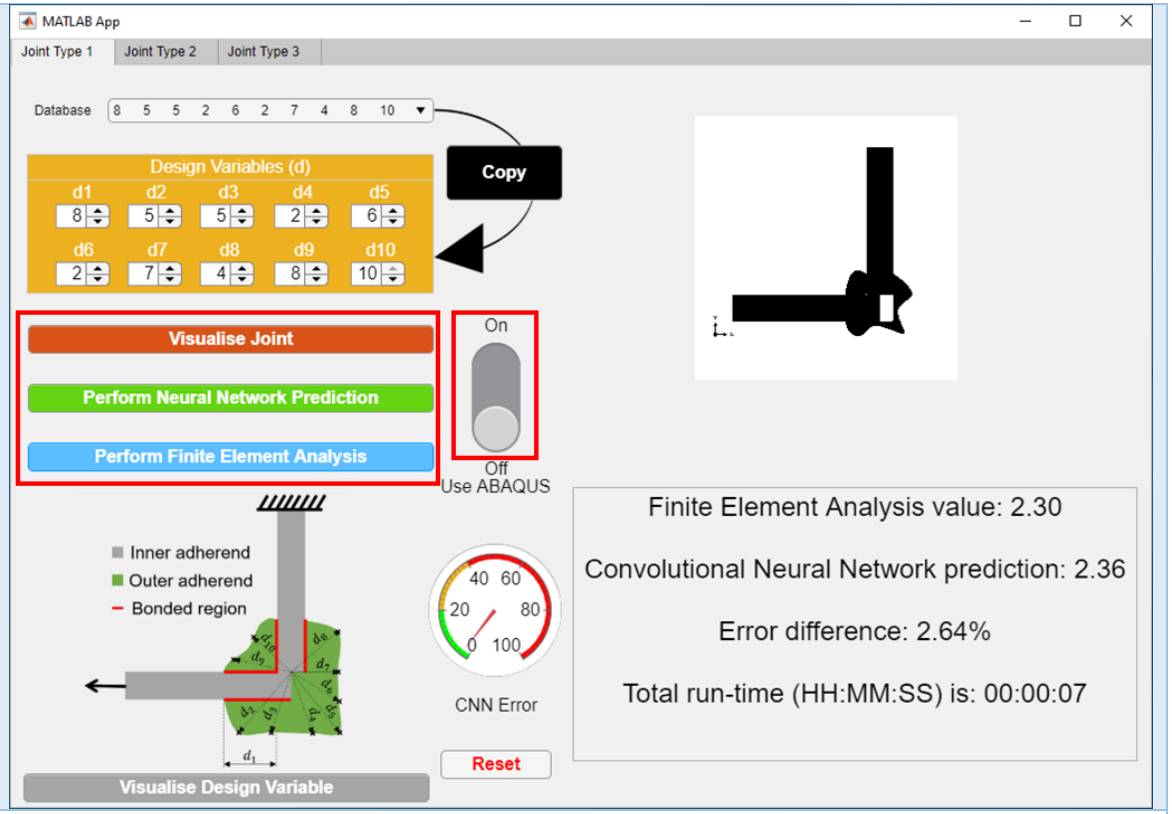
The user could also manually select a design variable.

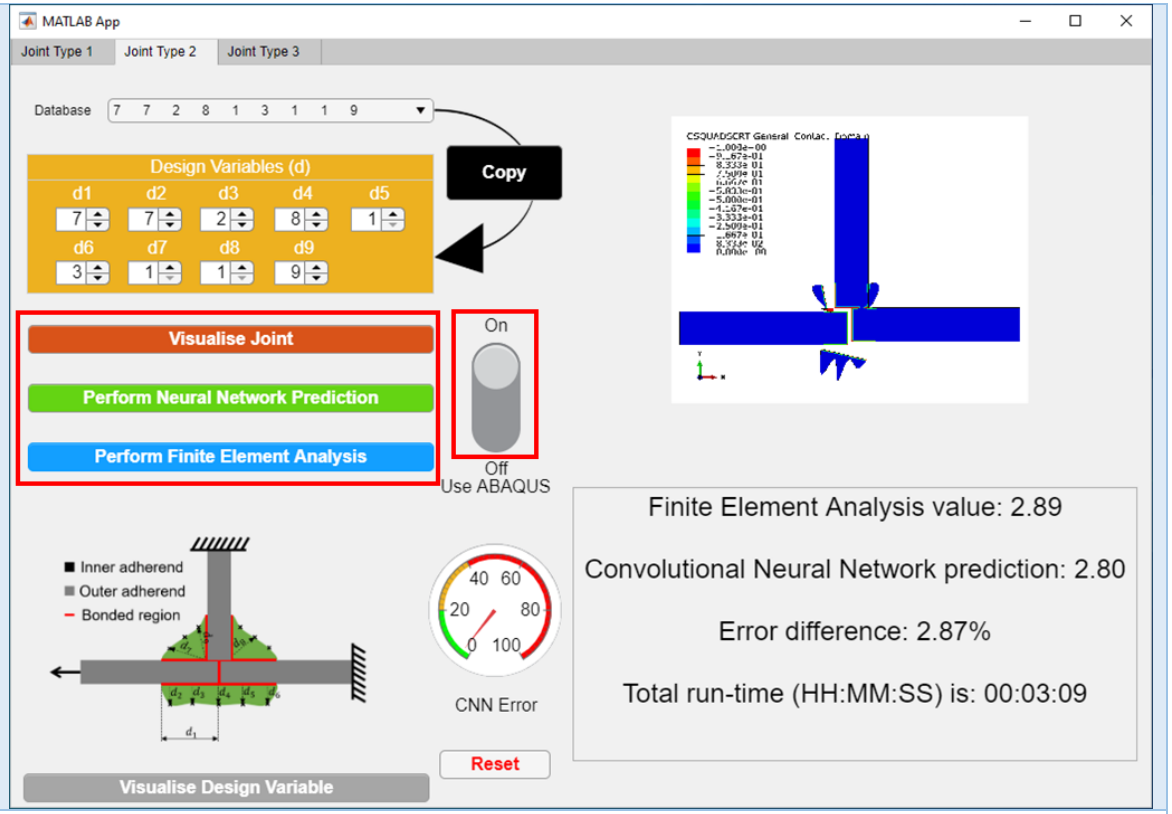


**Step 4.**

With the '***Use ABAQUS***' switch turned off; the three buttons would perform their respective task using the available database.

***Although the FEA would be copied from the saved database, the CNN prediction would be performed fully.***



| | | |
|---|---|---|
| **Step 5.** | Turning on the '***Use ABAQUS***' switch would result in the application ignoring the database and attempting to use the commercial product ABAQUS to perform the required task.<br><br>***An ABAQUS license would be required for this*** |  |