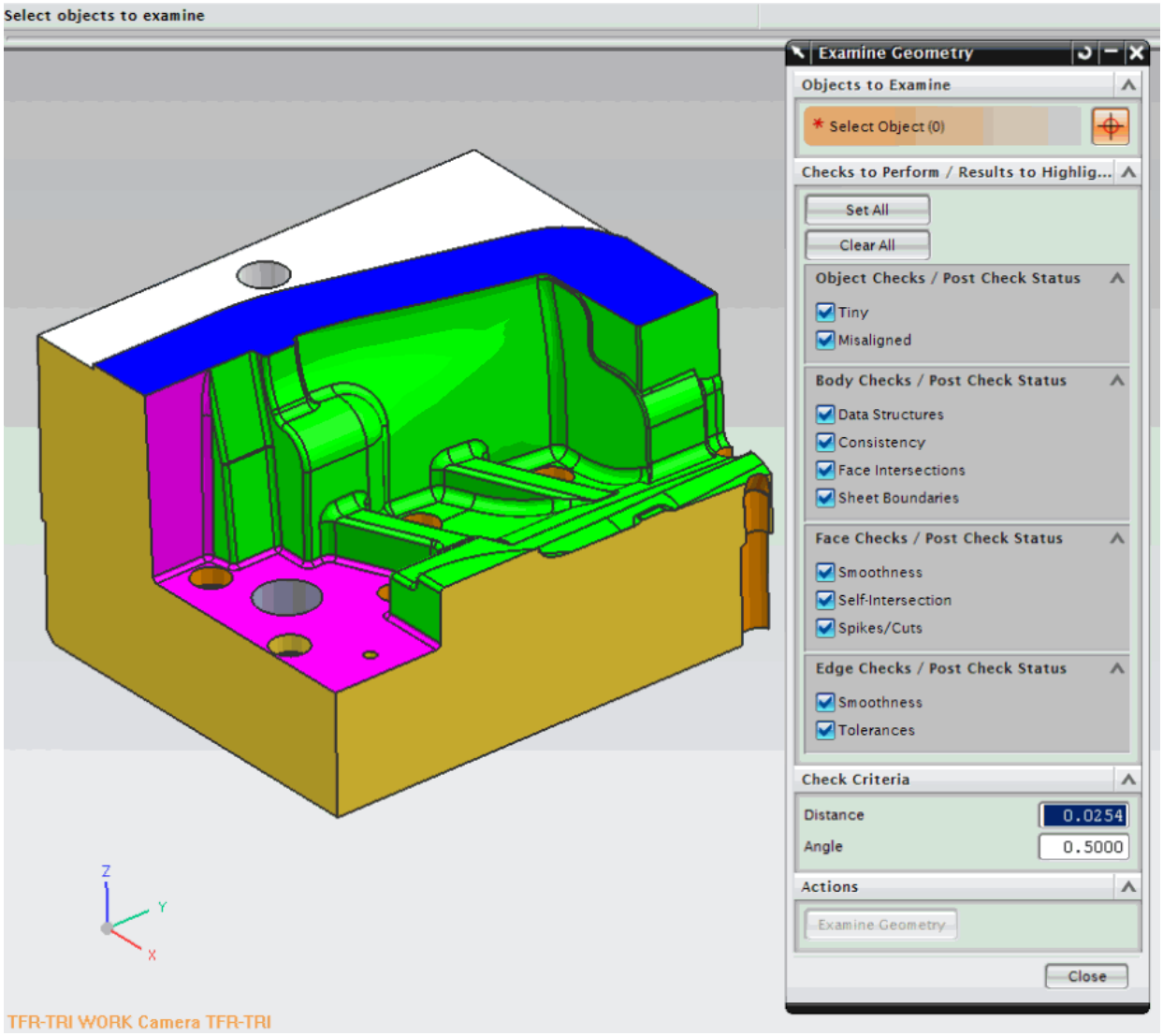
NX - Repairing errors found using the Examine Geometry function

During the creation and editing of **NX** part files, errors can be introduced into the structure of the part. Many **NX** users have developed procedures and standards that prevent the release and manufacturing of parts that do not pass ***Examine Geometry*** analysis. The ***Examine Geometry*** function allows users to locate these errors. These errors can then be repaired using various **Modeling Application** functions.

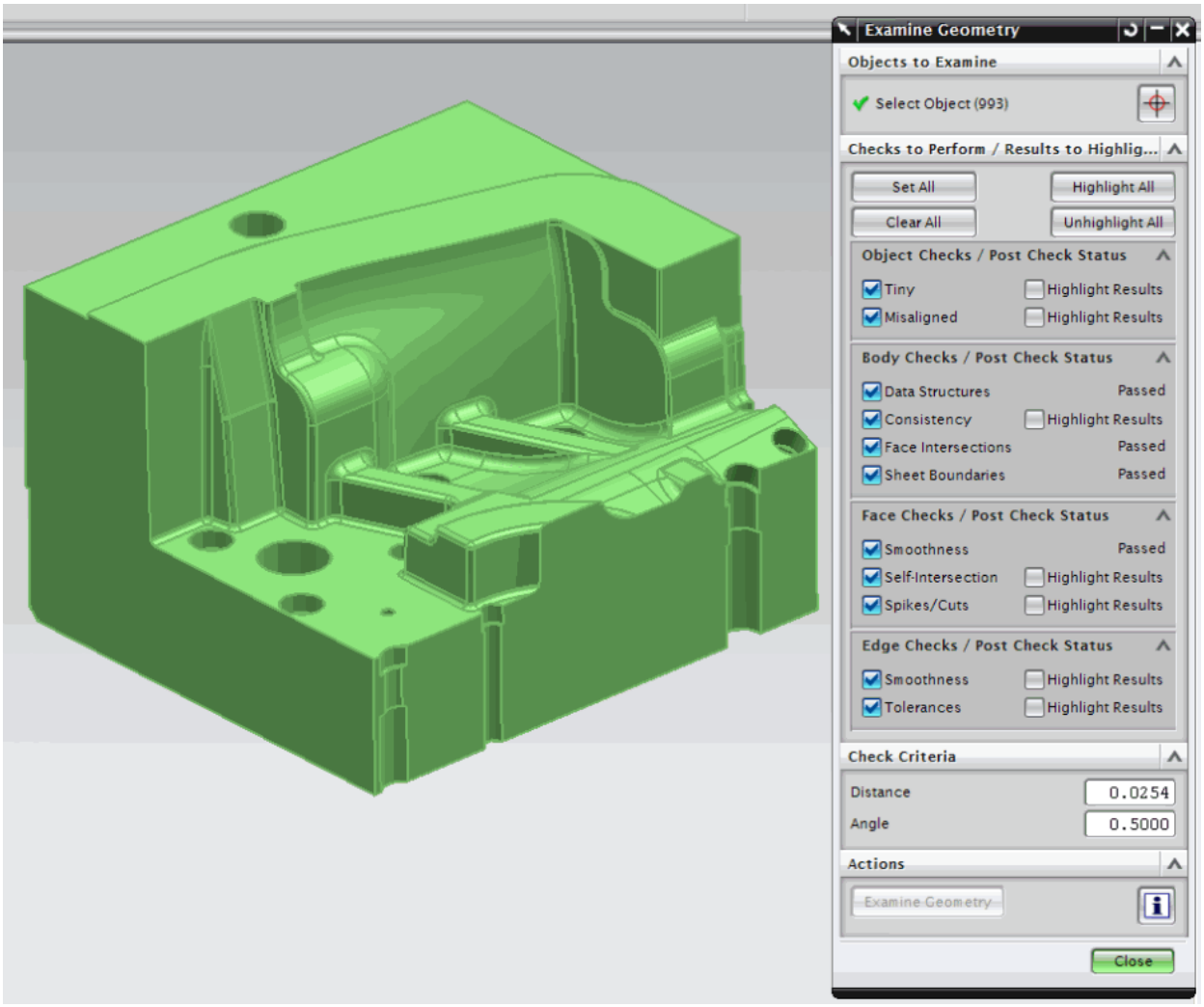
The first step is to find the error locations on the part. To do this, use ***Analysis => Examine Geometry.***



Select ***Set All*** to search for all available types of errors. Set the ***Distance*** and ***Angle*** **Check Criteria** to reasonable values. The values used are normally the same as the **Modeling Preferences Tolerance** values.

Next, use a window selection around the entire part. This will ensure that all objects in the part are chosen. Now select the ***Examine Geometry*** button.

The ***Examine Geometry*** function operates on the selected objects and the following screen is displayed:



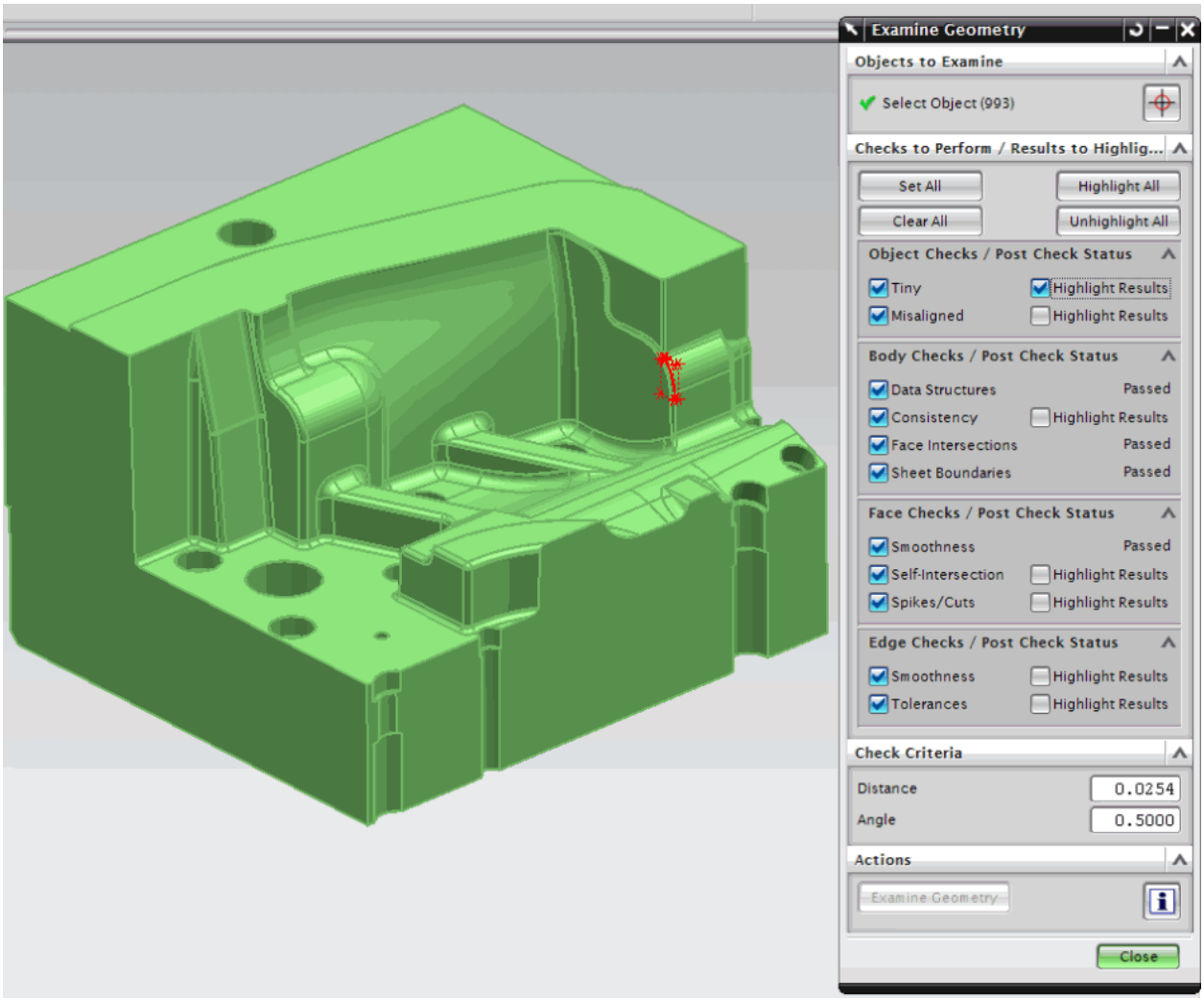
In this part, ***Examine Geometry*** found **Tiny** and **Misaligned** objects, **Consistency** errors, **Self Intersection** and **Spikes / Cuts** errors, and **Smoothness** and **Tolerances** errors.

To begin locating and repairing these errors select a **Highlight Results** checkbox next to one of the check titles. Note on the graphics screen that the offending objects are highlighted with asterisks and dotted lines bounding the errors. This highlighting allows users to zoom in to the area and begin the process of repair.

In general, the best practice is to locate and repair errors from one check at a time. Once the errors from the check have been repaired, rerun the ***Examine Geometry*** function. There are cases where repairing one type of error in the model will also repair other types of errors.

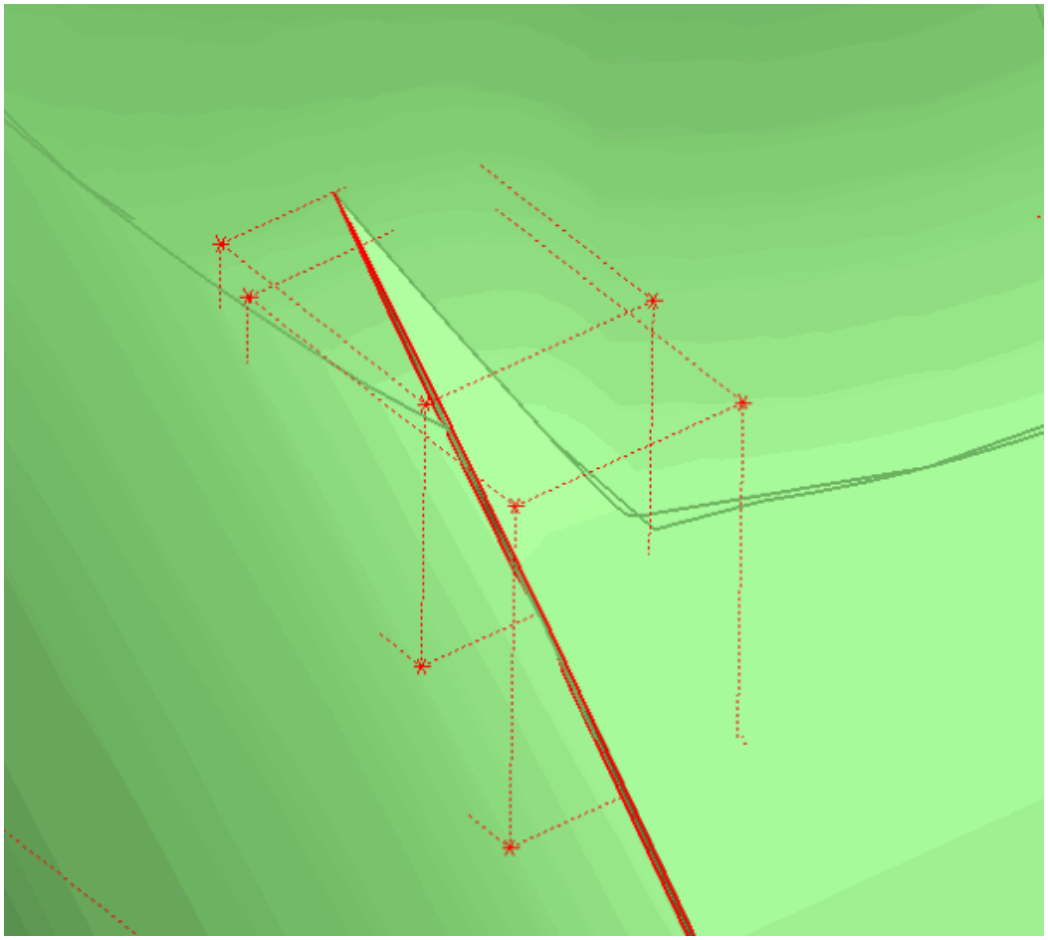
In this example, begin by choosing ***Highlight Results*** next to the ***Tiny*** object check.

Notice the red highlighted area on the model.



Zoom in to this area for a close visual inspection. Once zoomed in, checking and un-checking the ***Highlight Results*** box will alternately show and hide the red highlights. This is useful in the visual examination process. Using ***View => Operation => Regenerate Work*** will smooth the model edges on the display.

Note the zoomed view area of this model shows a ***Tiny*** object highlighted.



This view also shows that there are face edges which do not align correctly. These will display in the ***Misaligned*** check. At this point, we have identified and located areas of the model that are causing errors. The user must now decide what method to use to repair the problem areas.

In this case, the model is un-parameterized. There are various modeling methods available to repair this area. The user could create new tool bodies on the adjacent faces, then subtract them from the target to create new edges to apply blends. It would be more efficient to use ***Edit => Face => Delete Face*** to delete the failing faces and allow the recreation of these blended areas.

The ***Examine Geometry*** tool can be useful in finding errors and repairing part files. The tool will quickly isolate failing areas of a model. Using discretion in deciding which methods to use to repair the failed areas is important. Certain un-parameterized models are excellent candidates for the Face Editing tools in NX. Parameterized models usually require additional scrutiny to determine if features need to be revised, or may require a simple re-ordering of the features in the Part Navigator. With experience, users will be able to rapidly repair failed geometry.

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