

ASME Local Failure

ANSYS Mechanical Application

DOCUMENTATION

Extension version	201.4
Release date	13-Aug-21
Compatible ANSYS version	2020 R1



Table of Contents

1	INTRODUCTION	3
2	PRODUCT RESTRICTIONS	3
3	GETTING STARTED	4
3.1	INSTALLATION	4
3.2	LOADING EXTENSION.....	5
3.3	UNINSTALLING	6
4	ASME LOCAL FAILURE APP	8
4.1.1	<i>Background.....</i>	<i>8</i>
4.1.2	<i>Solution.....</i>	<i>8</i>
4.1.3	<i>Usage.....</i>	<i>8</i>
4.1.4	<i>Issues and Limitations</i>	<i>11</i>
5	REFERENCES	12
	APPENDIX A. EXAMPLES.....	13
	APPENDIX B. IMPLEMENTATION	22
	APPENDIX C. GENERAL ISSUES AND LIMITATIONS.....	23

1 Introduction

This application has been created using ACT (ANSYS Customization Toolkit), which provides an API (Application Programming Interface) to several ANSYS Workbench modules, including ANSYS Mechanical, ANSYS DesignModeler and ANSYS DesignXplorer. With ACT, the GUI (Graphical User Interface) of ANSYS Mechanical can be extended with new buttons. These buttons have customizable behaviour, which allows the developer to create new features, or re-use existing functionality that previously had to be included via APDL command objects. A few of the advantages with ACT include:

- Consistent handling of unit systems
- Manage user inputs
- Access to the Mechanical database as well as the ANSYS database and results
- Use Mechanicals graphics library for on-screen representation
- Shield the user from APDL code
- Create installable binary files, protecting intellectual property

2 Product Restrictions

The *ASME Local Failure* app described in this document utilize functionality in ANSYS that may or may not be available depending on the available ANSYS license level. The product restrictions are summarized in Table 2-1.

Table 2-1: Required ANSYS license level for the ASME Local Failure app

ANSYS license level (current)	Product code	Compatibility [Y/N]
DesignSpace	caewbpl3	N
Mechanical Pro	mech_1	N
Mechanical Premium	mech_2	Y
Mechanical Enterprise	ansys	Y
ANSYS license level (legacy)		
Professional NLT	prf	N
Professional NLS	prfnls	Y ¹
Structural	struct	Y

¹ The extension object is available. However, the multilinear hardening model is not available for this license level.

3 Getting Started

3.1 Installation

To install the extension, open ANSYS Workbench. On the project page, navigate to **ACT Start Page > Manage Extensions** and press the '+' icon to install a new extension (see Figure 3-1).

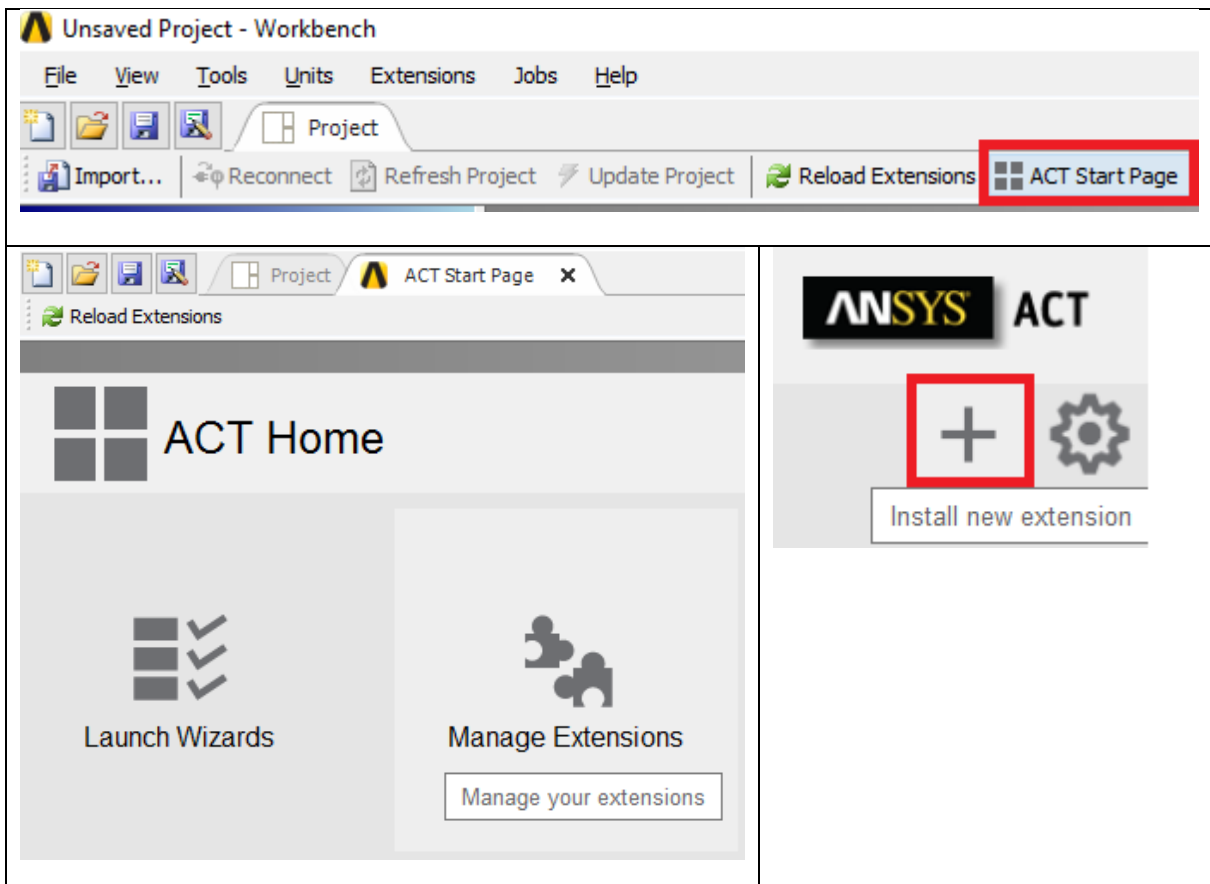


Figure 3-1: Navigating to the Manage Extensions page and installing a new extension

In the browsing window that appears, select the binary extension file (.wbex file) downloaded from ANSYS app store (see Figure 3-2).

Verify that the correct extension appears in the list in the *Extension Manager* by going to **ACT Start Page > Manage Extensions...** (see Figure 3-3).

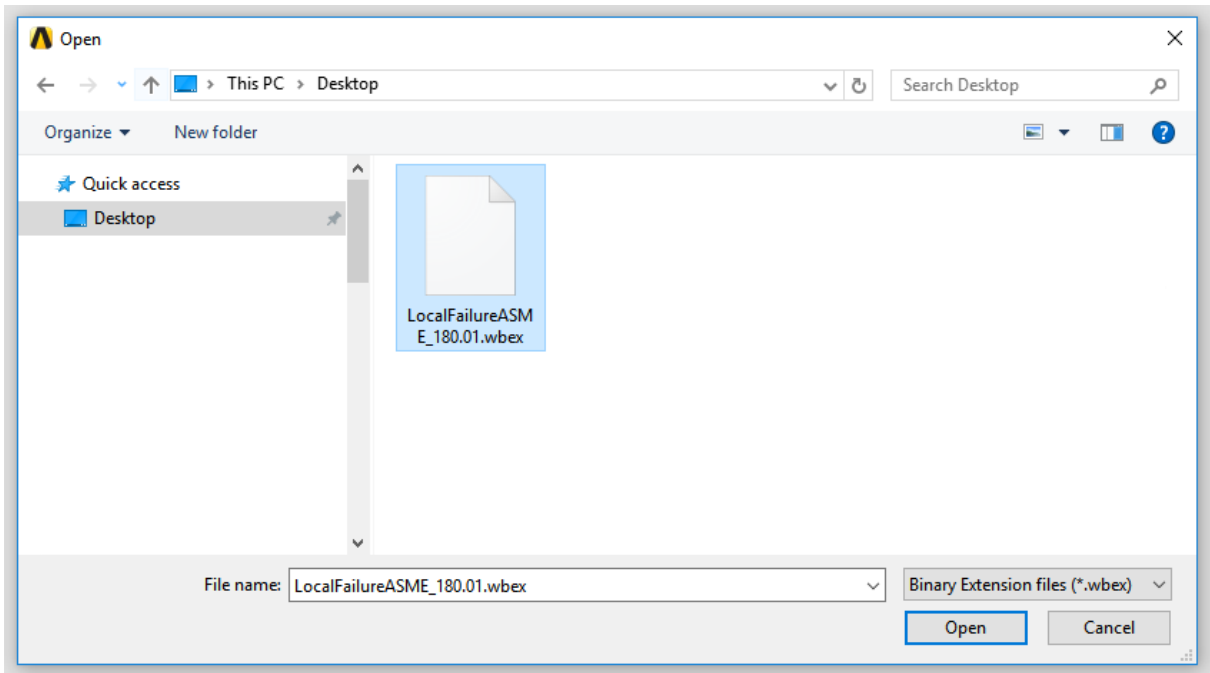


Figure 3-2: Select the .wbex file downloaded from ANSYS app store

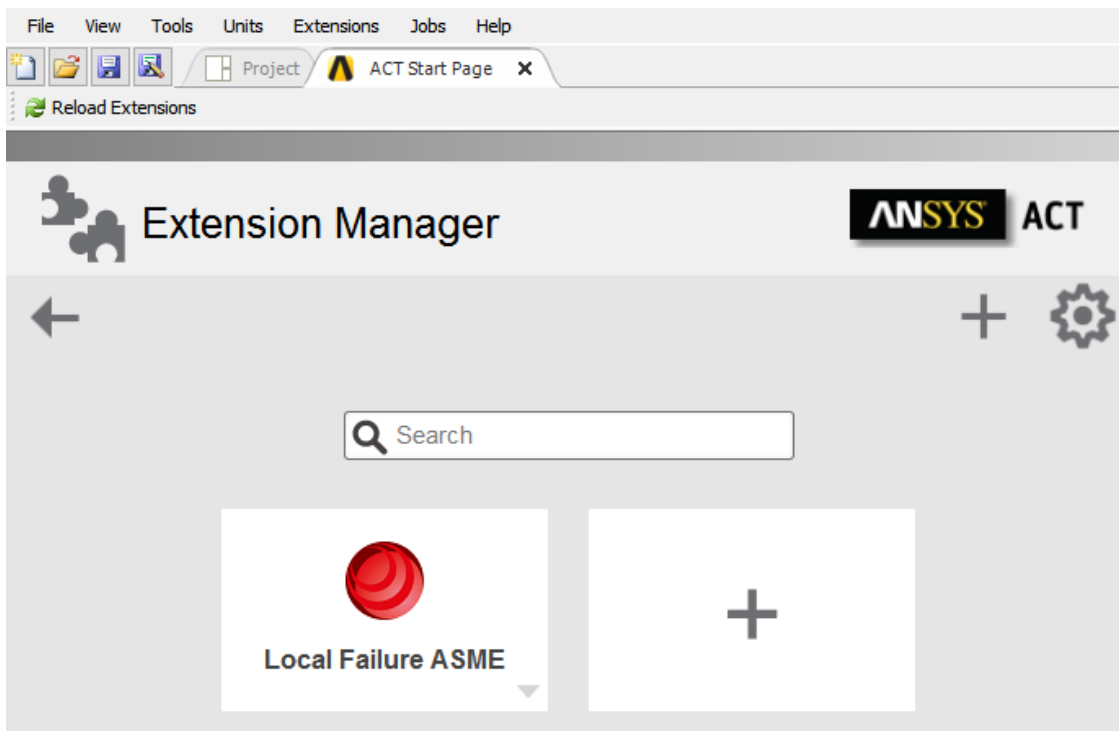


Figure 3-3 The newly installed extension should appear in the Extension Manager page

3.2 Loading Extension

To use the extension in a project, simply click on the extension on the *Extension Manager* page. The extension will highlight in green when properly loaded. Mechanical needs to be closed before the loading is effective, so close Mechanical if this is open.

Several extensions can be loaded into the same project. Once the project is saved with an extension, it will load automatically with the project, so this procedure only needs to be done once per project.

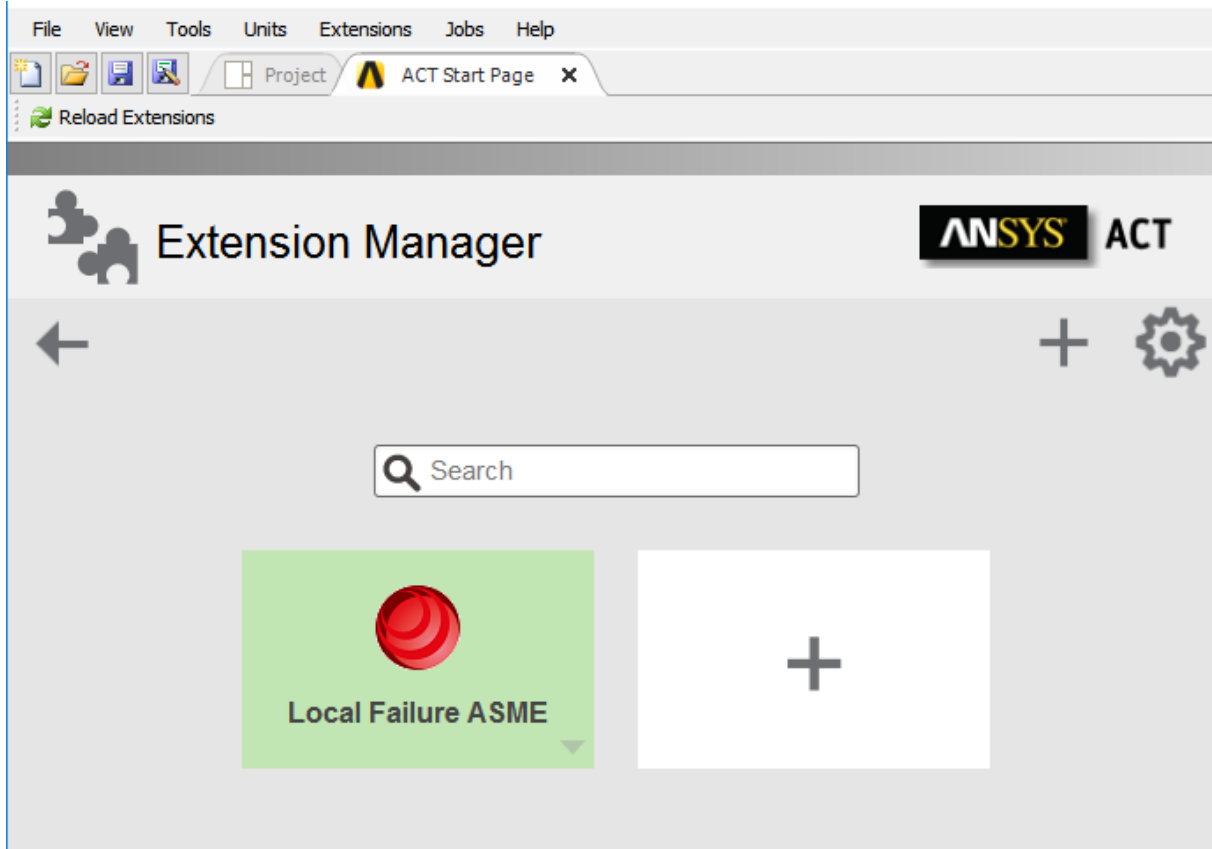


Figure 3-4: Properly loaded extension on the Extension Manager page

To unload the extension, open the *Extension Manager* page and click on the extension again. The extension is unloaded when the green highlighting disappears.

3.3 Uninstalling

To uninstall the extension, use the *Extension Manager* as described in section 3.1 and 3.2. From the drop-down menu on the bottom right of the extension, select *Uninstall*.

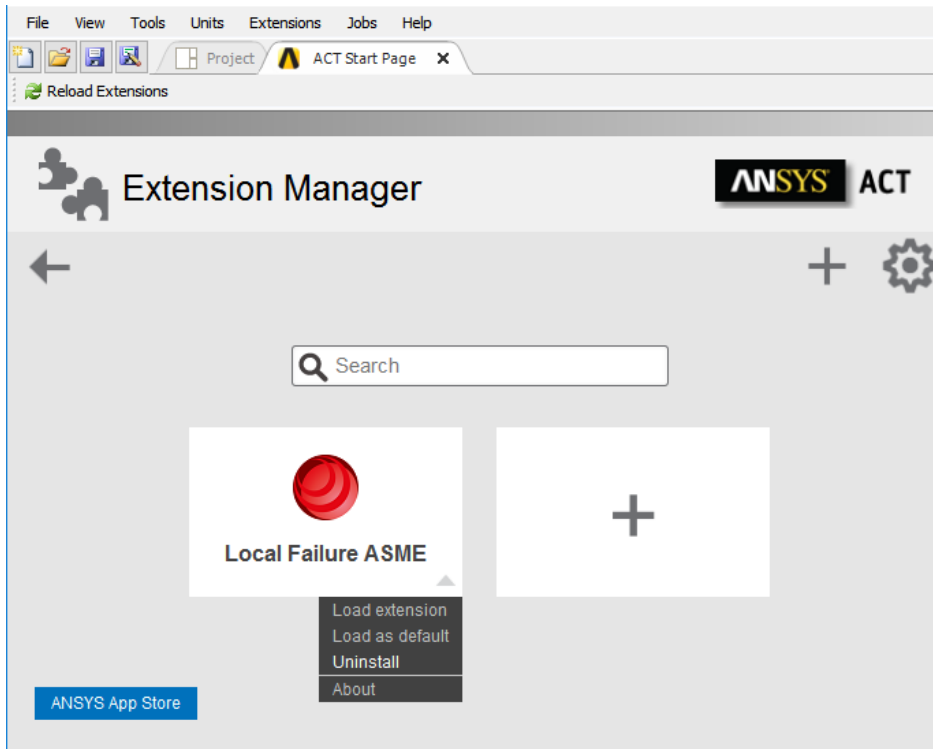


Figure 3-5: Uninstalling an extension from the Extension Manager page

4 ASME Local Failure app

4.1.1 Background

Local failure is a term used in ASME 2013 Sec. VIII Div.2 Part 5. (1) as a means to compare the magnitude of plastic strain values against limits set out by the code. The criterion uses triaxial stresses and material constants to establish a local failure strain. This strain is used together with the plastic strain and, if present, the forming strain, to calculate a utilization ratio against local failure.

Handling these stress and strain results, together with the material class dependant constants can be a tedious task, and easily prone to human error.

4.1.2 Solution

The Local Failure (ASME VIII Div.2) result object can be used as an efficient post-processing tool to plot the local failure utilization ratio according to ASME 2013 Sec. VIII Div.2 Part 5. section 5.3.3 *Elastic-Plastic Analysis* (1). The material properties are found for the specified material class according to ASME 2013 Sec. VIII Div.2 Table 5.7 (1). Furthermore, the Local Failure (ASME VIII Div.2) result object offers the possibility to specify forming strain.

4.1.3 Usage

The Local Failure (ASME VIII Div.2) result object can be applied in any *Static Structural* and *Transient Structural* analysis. Properties have to be selected or entered based on how the result shall be calculated. All properties are defined in Table 4-1.

The result can be scoped to any type of geometry, but not to mesh entities (directly or through Named Selections). The local failure utilization ratio according to ASME 2013 Sec. VIII Div.2 chap. 5.3.3.1 (1) or the strain limit damage according to ASME 2013 Sec. VIII Div.2 chap 5.3.3.2 (1) can be calculated. If the procedure *Loading Sequence* (ASME VIII 2-5.3.3.2) has been selected, the *Loading Sequence Table* has to be filled in.

The following material properties should at least be specified; *Material*, *Specified Minimum Yield Strength*, *SMYS* and *Specified Minimum UTS*, or defined by the user directly if *None (User Input)* is selected as *Material* property.

If the material assigned to the selected geometry has *Tensile Yield Strength* and/or *Tensile Ultimate Strength* defined in *Engineering Data*, the *Tensile Yield Strength* value will automatically be retrieved as *Specified Minimum Yield Strength*, *SMYS* and the *Tensile Ultimate Strength* value will automatically be retrieved as *Specified Minimum UTS* in the result detail view, or *Material Data Table*.

The material properties can be different for all bodies by selecting *No* for *Same Properties For All Bodies*. In the case of body dependent material properties, the *Material Data Table* and/or *Material Properties Table* have to be filled in (see Appendix C if the table does not pop up). Thus, it is not possible to select *No* if the geometry selection contains a single body.

If required, forming strain can be specified for a body selection.

The result plot shows the utilization ratio of equivalent plastic strain to the limiting triaxial strain UR_{LF} , as described in Appendix B. If the geometry contains shell elements, the selected shell location will be used as a uniform contour through the thickness. An example of property set and result is shown in Figure 4-1.

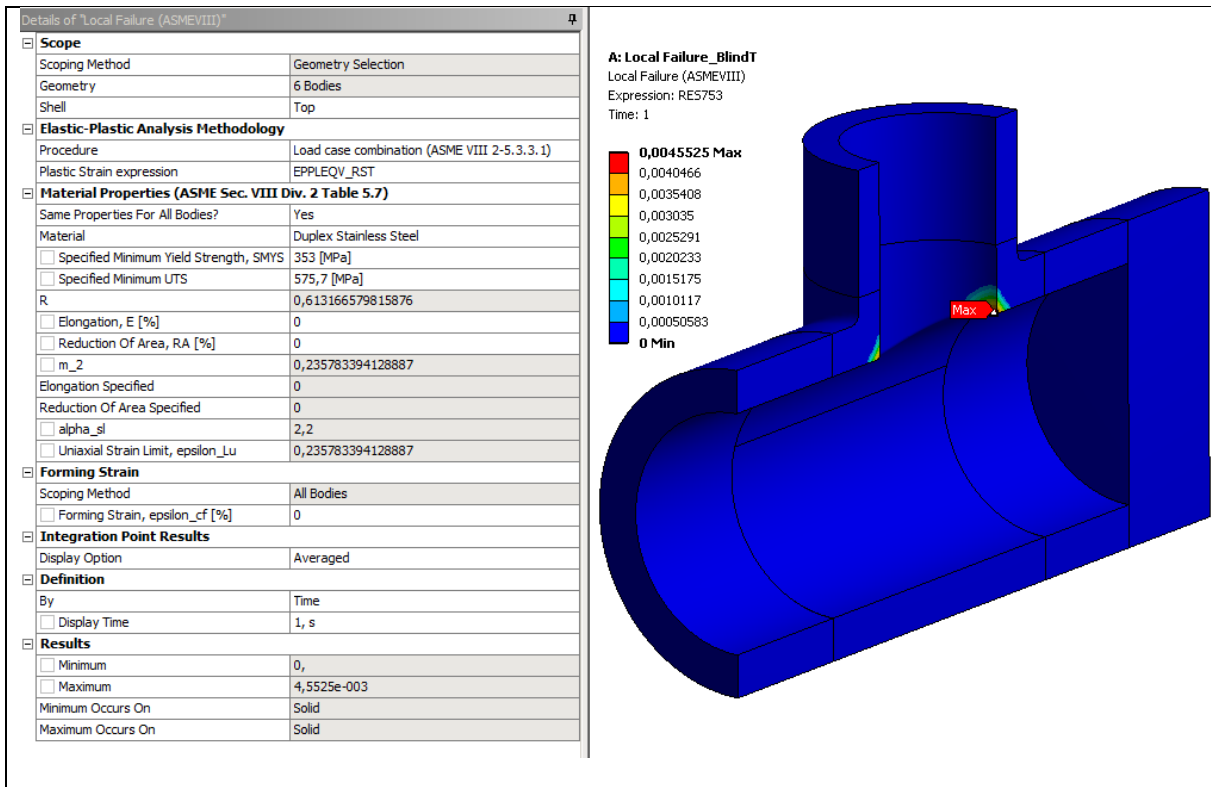


Figure 4-1: Details and graphic view of a Local Failure (ASME VIII Div.2) result object.

Table 4-1: Summary of properties from the Local Failure (ASME VIII Div.2) result object (grey cells are read-only properties).

Scope	
Geometry	Result geometry scope. Can be edges, vertices, faces or bodies
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> results are displayed for shell bodies
Elastic-Plastic Analysis Methodology	
Procedure	Specify the ASME procedure to be used whether <i>Load case combination (ASME VIII 2-5.3.3.1)</i> or <i>Loading Sequence (ASME VIII 2-5.3.3.2)</i>
Plastic Strain Expression	Plastic strain used for the calculation, can be specified for strain limit damage calculation only: EPPLEQV_RST (current equivalent plastic strain) or NLEPEQ (accumulated equivalent plastic strain)
Loading Sequence Table	Specified the loading sequence to be evaluated by giving the starting step and <i>load increments end steps</i> of the loading sequence.
Material Properties (ASME Sec. VIII Div. 2 Table 5.7)	

Same Properties For All Bodies	Specify whether the material properties are the same for all bodies in the geometry scope or not
Material	Specify, for the entire geometry scope, which Material Class from ASME Sec. VIII Div. 2 Table 5.7 is used to calculate the material property values, or if the material property values are defined as user input
Specified Minimum Yield Strength, SMYS	<i>Specified Minimum Yield Strength</i> to be used, for the entire geometry scope
Specified Minimum UTS	<i>Specified Minimum Ultimate Tensile Strength</i> to be used, for the entire geometry scope
R	Ratio of the <i>Specified Minimum Yield Strength</i> to the <i>Specified Minimum Ultimate Tensile Strength</i>
Elongation, E [%]	% elongation determined from the applicable material specification (optional, see ASME Sec. VIII Div. 2 Table 5.7) (for the entire geometry scope)
Reduction Of Area, RA [%]	% reduction in area from the applicable material specification (optional, see ASME Sec. VIII Div. 2 Table 5.7) (for the entire geometry scope)
m_2	Calculated using ASME Sec. VIII Div. 2 Table 5.7 formula, or defined as a user input. Scoped to the entire geometry
Elongation Specified	<i>Elongation Specified</i> calculated using ASME Sec. VIII Div. 2 Table 5.7 formula (for the entire geometry scope)
Reduction Of Area Specified	<i>Reduction of Area Specified</i> calculated using ASME Sec. VIII Div. 2 Table 5.7 formula (for the entire geometry scope)
α_{sl}	α_{sl} defined using ASME Sec. VIII Div. 2 Table 5.7 or defined as a user input (for the entire geometry scope)
Uniaxial Strain Limit, ϵ_{Lu}	ϵ_{Lu} defined using ASME Sec. VIII Div. 2 Table 5.7 or defined as a user input (for the entire geometry scope)
Material Data Table	Same properties as previously defined by body scope (see Appendix C for body ID description)
Material Properties Table	Same properties as previously defined by body scope (see Appendix C0 for body ID description)
Forming Strain	
Geometry	Forming strain scope, can be bodies only (optional, see ASME Sec. VIII Div. 2 section 5.3.3 Step 4)
Forming Strain, ϵ_{cf} [%]	% forming strain to be used for the forming strain body scope (default 0, see ASME Sec. VIII Div. 2 section 5.3.3 Step 4)
Integration Point Results	
Display Option	Specify whether the result are displayed as <i>Averaged</i> (default) or <i>Unaveraged</i> result
Definition	
By	Select whether to view the results at a specific time or a specific solution set

Display Time	Time point at which to display the result (see limitations in Appendix C).
Substep	Which substep to view the result from

4.1.4 Issues and Limitations

The Local Failure (ASME VIII Div.2) result object is limited to *Static Structural* and *Transient Structural* analyses, 3D geometries and solid or surface bodies.

Results cannot be displayed by *Maximum Over Time* or *Time Of Maximum*.

The geometry scope must have an assigned material with plastic behaviour defined to be valid.

5 References

1. ASME. *Section VIII, Division 2*. 2013.

Appendix A. Examples

This example shows, step by step, how to use the Local Failure (ASME VIII Div.2) result object, as well as verification of the results.

A *Static Structural* analysis is created and a Duplex Stainless Steel material with elastic-plastic behavior is used. The material properties are summarized in Table 5-1 and Figure 5-1.

Table 5-1: Material Properties of the Duplex Stainless Steel material.

Material Property		Value
Elastic Modulus	[GPa]	185.7
Poisson's ratio	[-]	0.31

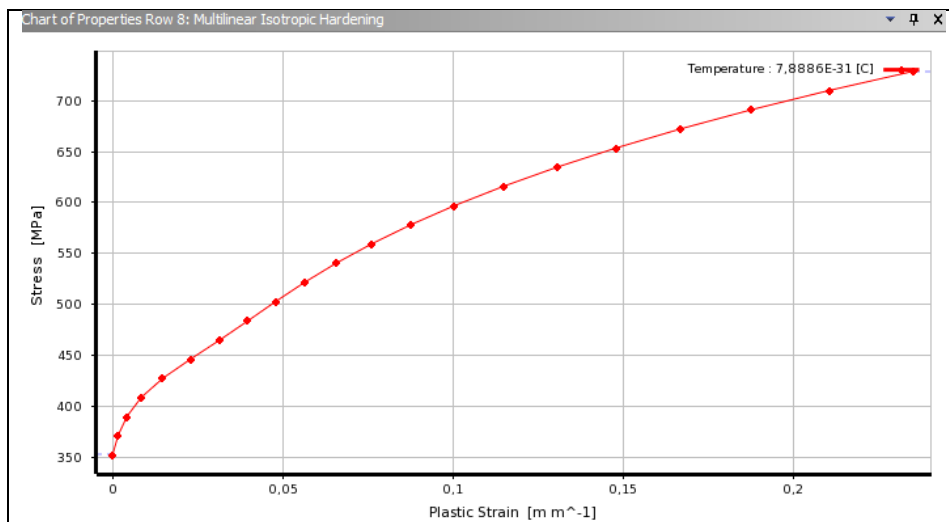


Figure 5-1: Multilinear Isotropic Hardening 'Stress - Plastic Strain' curve of the Duplex Stainless Steel material

The geometry used in this example is a Blind Tee, modeled as a solid multibody part. The previously defined material is assigned to all bodies.

The model is subjected to an internal pressure with its corresponding cap force. Further, the outlet surface is fixed in the axial and tangential direction with respect to a local coordinate system as shown in Figure 5-2. For simplicity, symmetry is utilized such that only half of the geometry model is used.

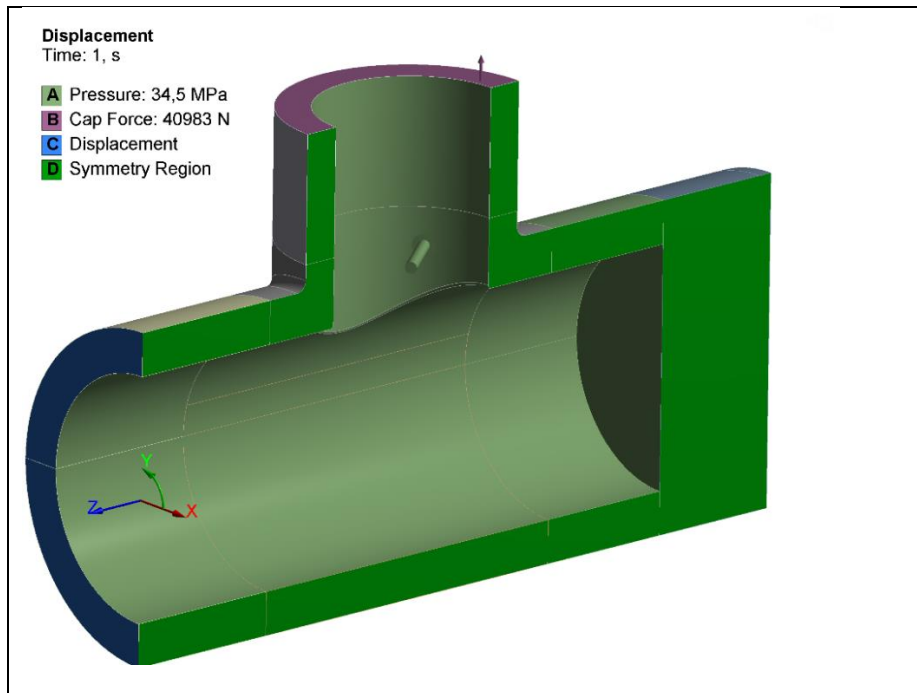


Figure 5-2: Loads and boundary conditions for the local failure example model

To evaluate the local failure utilization in the blind tee, a Local Failure (ASME VIII Div.2) result object is added to the structure tree and scoped to all bodies. This object is found in the *Results* drop-down menu of the extension, as seen in Figure 5-3.

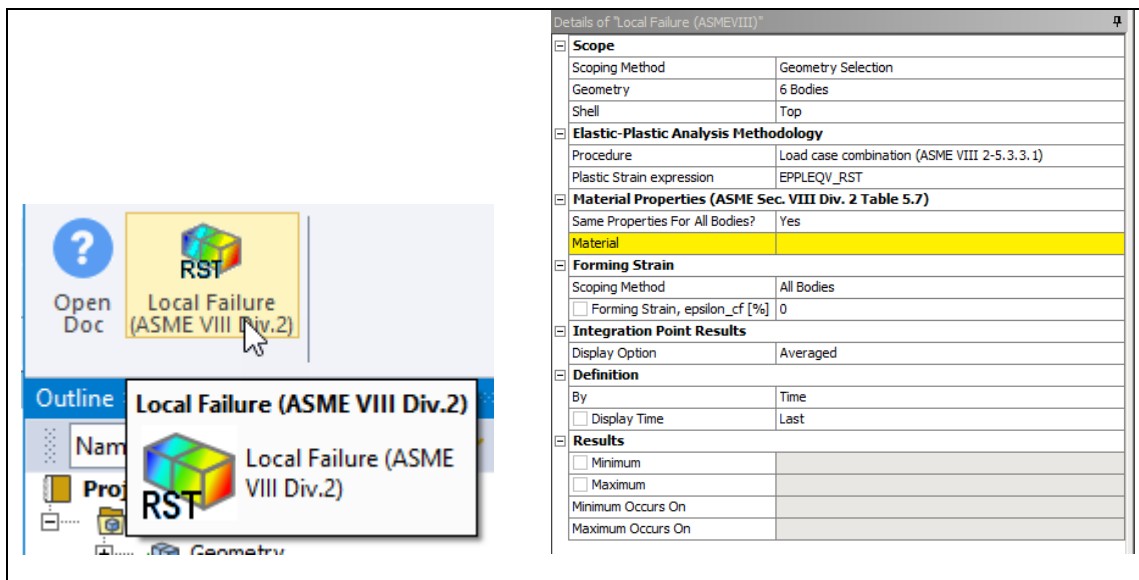


Figure 5-3: Inserting a Local Failure (ASME VIII Div.2) object and Details of Local Failure (ASME VIII Div.2) result object after insertion

When the material class *Duplex Stainless Steel* is selected, two new fields appear as shown in Figure 5-4.

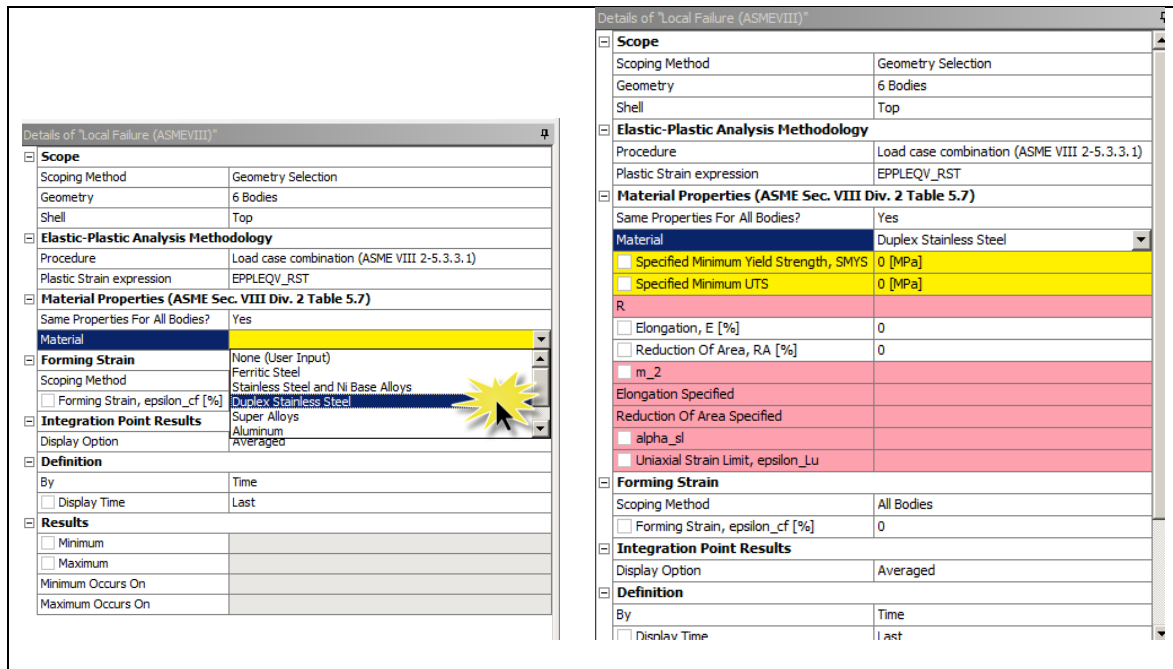


Figure 5-4: Details of Local Failure (ASME VIII Div.2) result object, before and after selection of a material class.

The *Specified Minimum Yield Strength, SMYS* is set to 353 MPa and the *Specified Minimum UTS* to 575.7 MPa as shown in Figure 5-5. After defining both properties, the result object changes status from invalid to valid, as a sufficient number of properties have been filled in to define the *Uniaxial Strain Limit* ϵ_{Lu} , α_{sl} and m_2 .

Note that:

- It is not required to define values for *Elongation, E [%]* and *Reduction of Area, RA [%]*, and in this example they are kept to 0. In cases where their values are modified, related properties are updated automatically.
- If setting *Material* to *None (User Input)*: *Uniaxial Strain Limit* ϵ_{Lu} , α_{sl} and m_2 properties appear as editable, and to be specified by the user, as illustrated in Figure 5-6.
- It is possible to define the *Tensile Yield Strength* and *Tensile Ultimate Strength* of the material in *Engineering Data*, as shown in Figure 5-7. Their values are then directly retrieved as *Specified Minimum Yield Strength, SMYS* and *Specified Minimum UTS* in the result object.

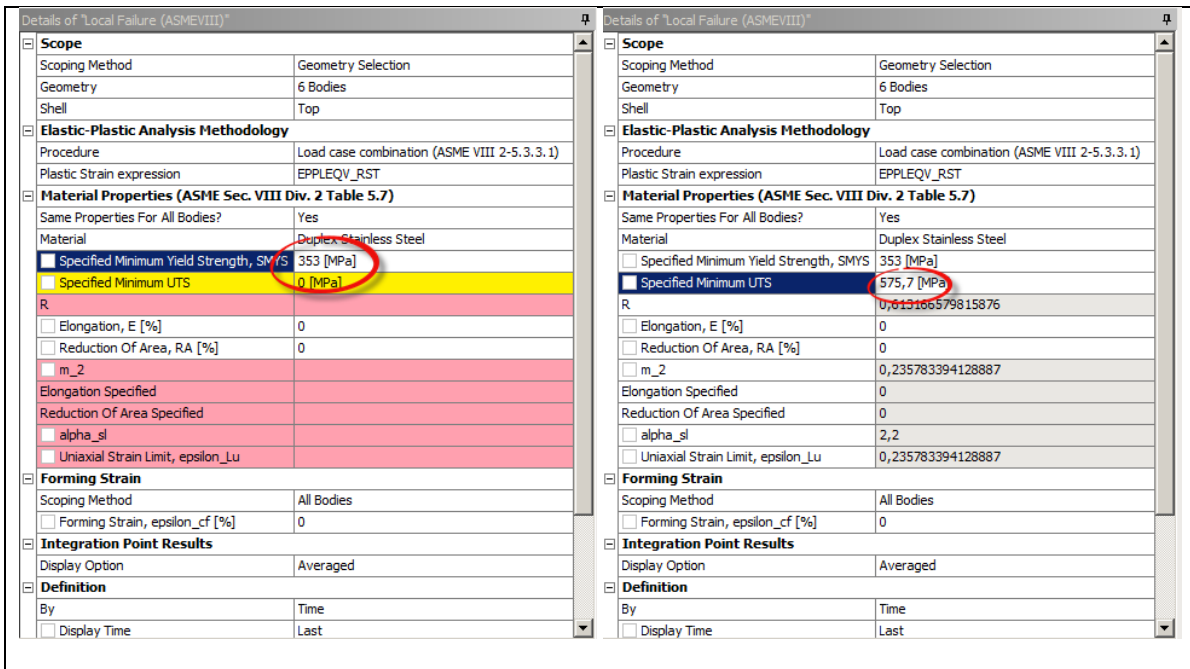


Figure 5-5: Details of Local Failure (ASME VIII Div.2) result object, after defining the Specified Minimum Yield Strength, SMYS and after defining the Specified Minimum UTS.

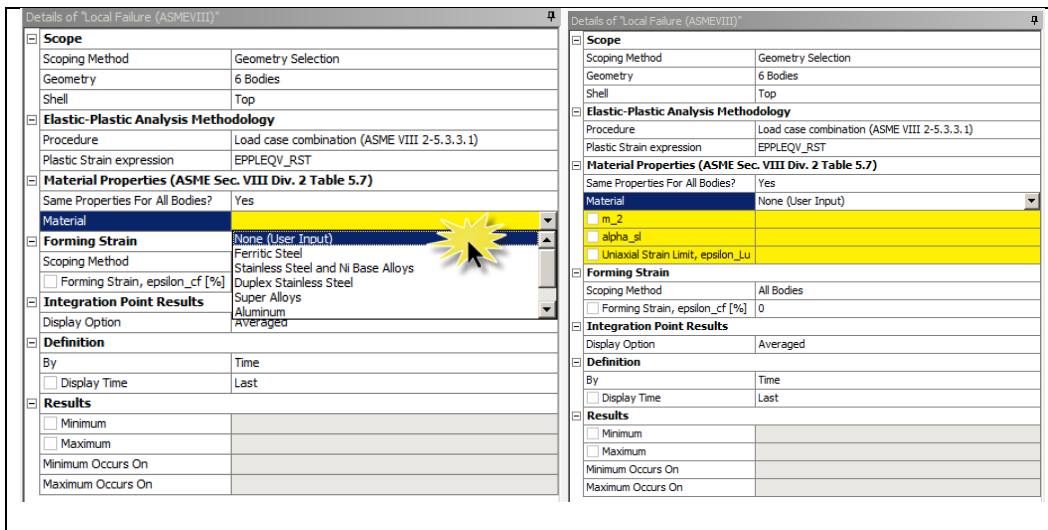


Figure 5-6: Details of Local Failure (ASME VIII Div.2) result object, before and after selection of None (User Input) instead of material class as Material.

	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				
8	Multilinear Isotropic Hardening	Tabular			
11	Tensile Yield Strength	353	MPa		
12	Tensile Ultimate Strength	575,7	MPa		

Figure 5-7: Tensile Yield Strength and Tensile Ultimate Strength defined in Engineering Data as Specified Minimum Yield Strength, SMYS and the Specified Minimum UTS respectively.

The result object is evaluated, and the corresponding plot is shown in Figure 5-8.

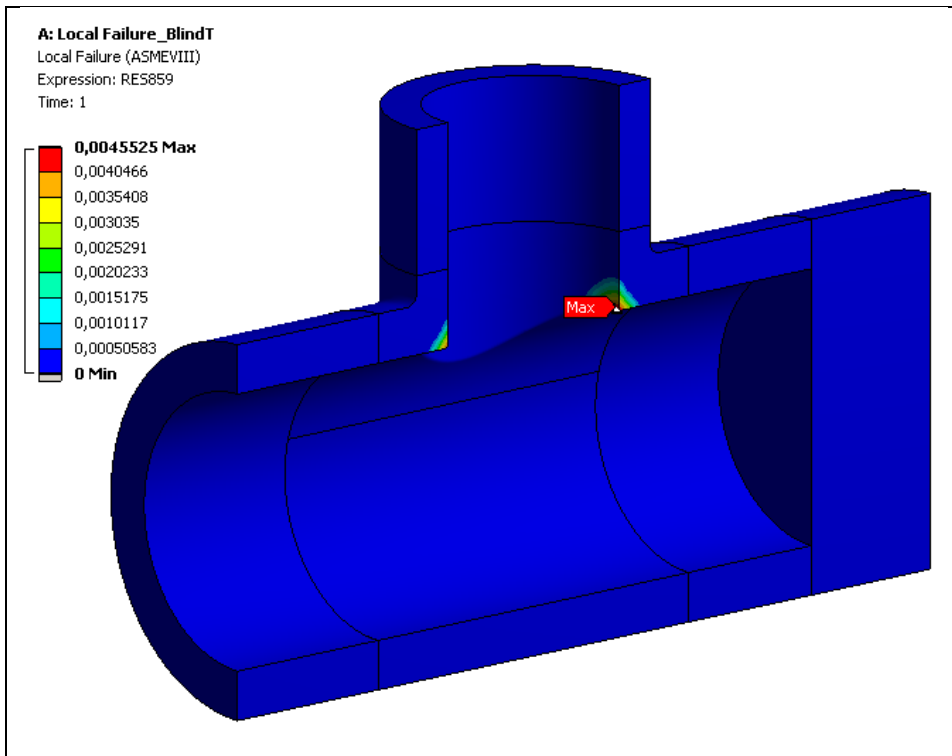


Figure 5-8: Results object plot after evaluation (no forming strain).

A second result is created by duplicating the first one, and 0.2 % forming strain is specified to the green/selected body as shown in Figure 5-9 and Figure 5-10.

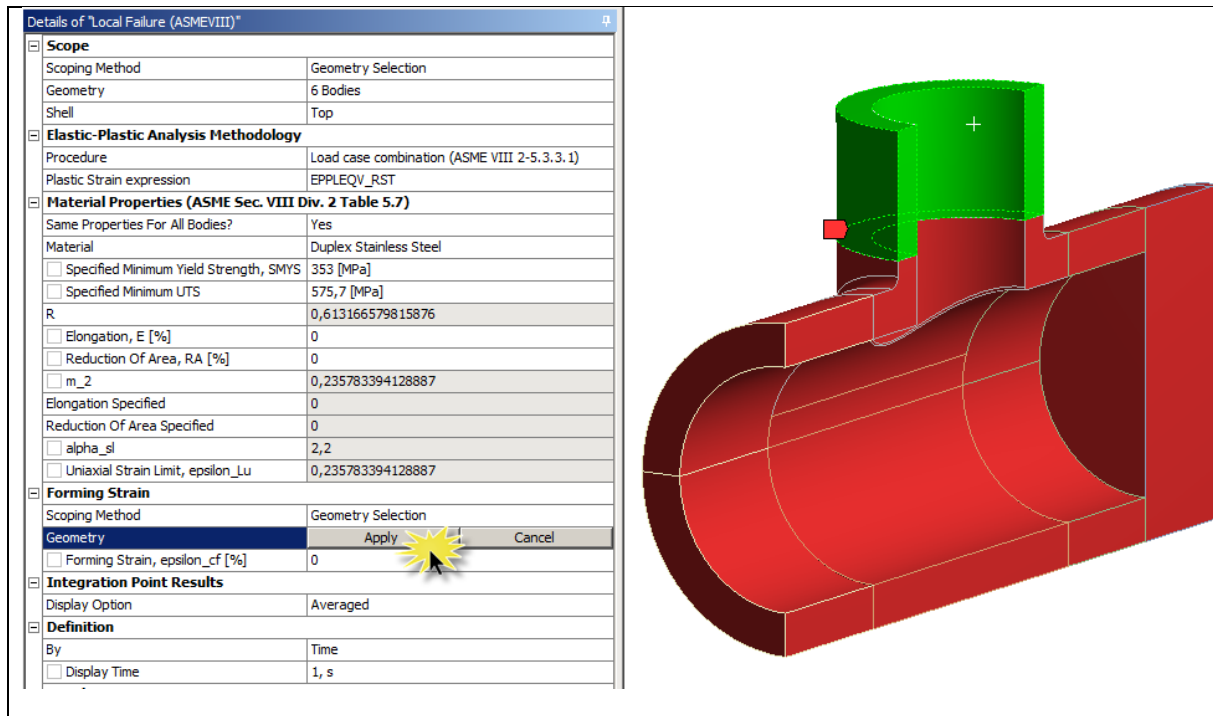


Figure 5-9: Body selection for forming strain.

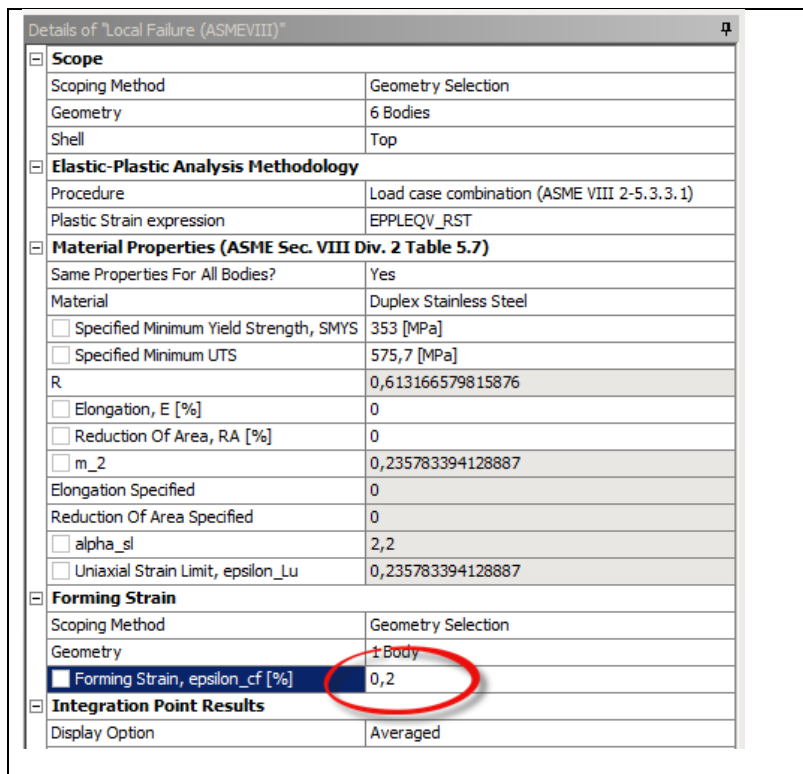


Figure 5-10: Details of Local Failure (ASME VIII Div.2) result object including forming strain

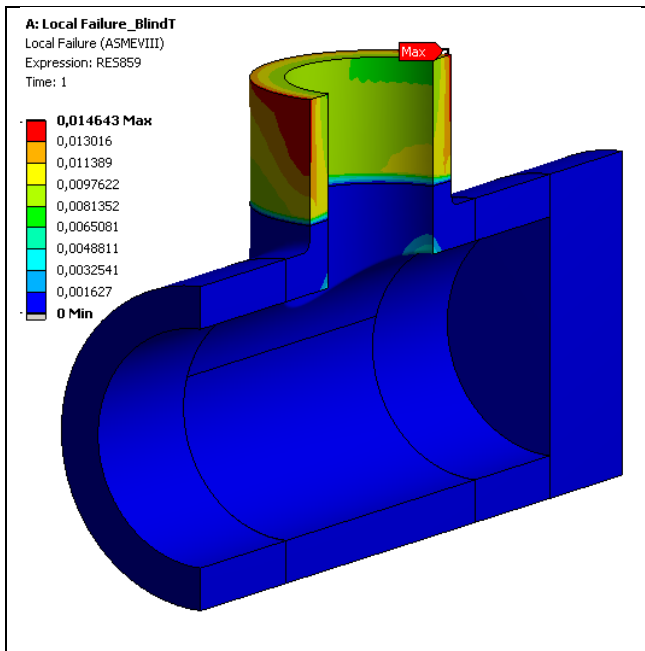


Figure 5-11: Results object plot after evaluation (with forming strain)

Verification of the result at the node with maximum utilization:

The material properties *Uniaxial Strain Limit* ϵ_{Lu} , α_{sl} and m_2 are extracted from Table 5.7 in (1) with Specified *Minimum Yield Strength* equal to 353 MPa and a *Specified Minimum UTS* equal to 575.7 MPa. The material properties are the same as evaluated through the result object, as shown in Figure 5-5: $\epsilon_{Lu} = m_2 = 0.2358$ and $\alpha_{sl} = 2.2$.

To verify the result, the equivalent stress and principal stresses are extracted at the node with maximum utilization, see Figure 5-12. Those results, used in combination with the material properties in equation 5.6 in (1), give the limiting triaxial strain. The ratio obtained by hand calculations below is equal to the one obtained with the extension result object in Figure 5-8.

$\frac{\sigma_1 + \sigma_2 + \sigma_3}{3\sigma_e} = 0.2471$	<p>From equation 5.6 in (1):</p> $\epsilon_L = 0.2750$ $\epsilon_{peq} = 0.001251$ $\epsilon_{cf} = 0$	$UR_{LF} = \frac{\epsilon_{peq} + \epsilon_{cf}}{\epsilon_L} = 0.004552$
$\epsilon_{Lu} = m_2 = 0.2358$		
$\alpha_{sl} = 2.2$		

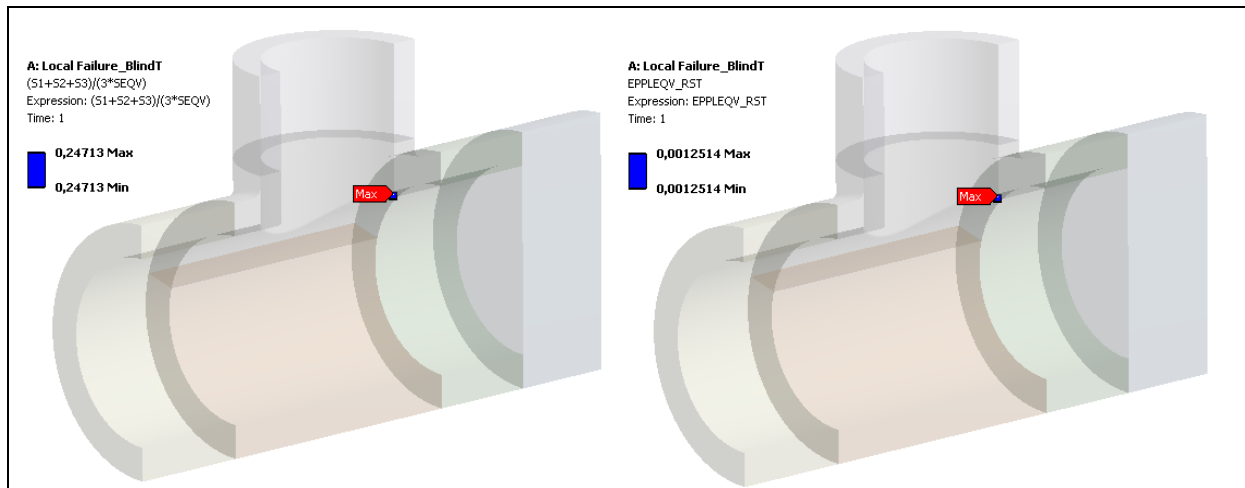


Figure 5-12: ANSYS sum of Principal Stresses over 3 times the Von Mises equivalent stress and Equivalent Plastic Strain results for result verification (no forming strain).

Verification of the result at the node with maximum utilization (result object with forming strain):

To verify the result obtained with forming strain, the equivalent stress and principal stresses are extracted at the node with maximum utilization, see Figure 5-13. Those results, used in combination with the material properties in equation 5.6 in (1), give the limiting triaxial strain. The ratio obtained by hand calculations below is equal to the one obtained with the extension result object Figure 5-11.

$\frac{\sigma_1 + \sigma_2 + \sigma_3}{3\sigma_e} = 0.6400$	From equation 5.6 in (1):	$\varepsilon_L = 0.1366$	$UR_{LF} = \frac{\varepsilon_{peq} + \varepsilon_{cf}}{\varepsilon_L} = 0.014642$
$\varepsilon_{Lu} = m_2 = 0.2358$			
$\alpha_{sl} = 2.2$			
$\varepsilon_{peq} = 0$			
$\varepsilon_{cf} = 0.002$			

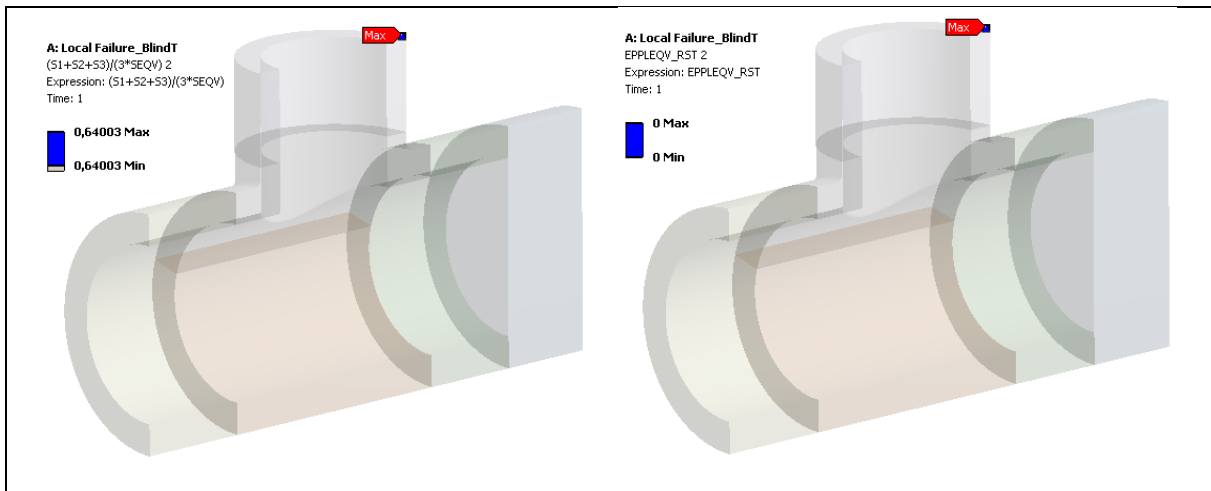


Figure 5-13: ANSYS sum of Principal Stresses over 3 times the Von Mises equivalent stress and Equivalent Plastic Strain results for result verification (result object including forming strain).

Appendix B. Implementation

If *Load Case Combination (ASME VIII 2-5.3.3.1)* is specified as Procedure, the limiting triaxial strain is calculated according to Eq. 5.6 in (1) as described below:

- ε_{LW} , α_{sl} and m_2 : defined/calculated using Table 5.7 in (1)
- σ_e , σ_1 , σ_2 and σ_3 : equivalent stress and principal stresses are calculated by ANSYS
- ε_{peq} : plastic strain calculated by ANSYS using the specified plastic strain expression EPPLEQV or NLEPEQ.
- ε_{cf} : when required, see section 5.3.3 Step 4 in (1), forming strain defined by the user.

The local failure utilization is found as UR_{LF} from the equation 5.7 in (1):

$$UR_{LF} = \frac{\varepsilon_{peq} + \varepsilon_{cf}}{\varepsilon_L} \quad \text{Equation 5-1}$$

If *Loading Sequence (ASME VIII 2-5.3.3.2)* is specified as Procedure, the strain limit damage is calculated according to Eq. 5.8, 5.9, 5.10 and 5.11 in (1) with the following ANSYS results:

- ε_{LW} , α_{sl} and m_2 : defined/calculated using Table 5.7 in (1)
- $\sigma_{e,k}$, $\sigma_{1,k}$, $\sigma_{2,k}$ and $\sigma_{3,k}$: equivalent stress and principal stresses are calculated by ANSYS for each step k specified in *Loading Sequence Table*.
- $\Delta\varepsilon_{peq,k}$ are calculated by Ansys at each step k as the difference of accumulated plastic strain (NLEPEQ) : $\Delta\varepsilon_{peq,k} = \varepsilon_{peq,k} - \varepsilon_{peq,k-1}$
- ε_{peq} : equivalent plastic strain calculated by ANSYS
- ε_{cf} : when required, see section 5.3.3 Step 4 in (1), forming strain defined by the user.

ε_{LW} , α_{sl} and m_2 properties can be the same for the entire geometry scope or body dependent. Forming strain can be added for the forming strain geometry scope only.

For averaged results, the utilization is calculated at nodes from the nodal averaged stress components. The component values at midside nodes are the average of the two neighbour corner nodes averaged component values. Results are averaged across bodies.

For unaveraged results, results are plotted at element nodes. The result value at midside nodes is the average of the two neighbouring corner nodes results.

Appendix C. General Issues and limitations

One issue regarding using results from another analysis system in load case definition tables: When this is used, and that other analysis system is invalidated/rerun, the result object is not immediately invalidated. Only when the result item is clicked once, the result item is invalidated.

Stopping the results evaluation by clicking “Stop Solution” at the ANSYS Workbench Solution Status pop-up window is not possible.

The *Display Time* is not working correctly in the case that the time specified does not have results stored. To ensure expected results, only select points in time where results are stored. In additions, for some result items the *Display Time* should not be used at all. This is specified in the Usage chapter for the affected result items.


Faces and edges that have been created as *Virtual Topology* are not selectable as geometry input.

Duplicating an evaluated result item results in an un-evaluated result item. The duplicate result item needs to be re-evaluated.

Three issues are related to pop-up tables. In the case that Mechanical is open in a display other than the main display, the table may not pop up. In the case that Mechanical is open in the main display, but the Mechanical window is scaled too small, the table may not be visible. The only way to close the table is to click on “Cancel” or “Apply”, clicking on the top-right red cross will not close the table.

When inserting an item under the *Setup* level (Initial Conditions, Loads or Supports), and then un-loading the extension, the item is not possible to delete afterwards. The extension needs to be reloaded before the item can be deleted.

For result items, the availability to select geometry as mesh elements or nodes is limited.

Some load and result objects refer to *Body ID*. The *Body ID* can be found by using *Selection Information*  in the main toolbar, as shown in Figure 5-14. *ID (beta)* displayed under definition in the detail view of a body, is not the *Body ID*.

Selection Information												
Coordinate System:		Global Coordinate System		Show Individual								
Entity	Volume (mm ³)	Mass (kg)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Nodes	Elements	Body	Type	TOPOID	PARTID	REFERENCE_ID
Body 1	59.42	4.6407e-004	-0.10183	1.3947	-4.4229	4398	2519	Solid	Solid	3221225473	22	23
Body 2	127.75	9.9773e-004	-1.4473	0.82476	9.1197	3129	1862	Solid	Solid	3221225474	22	54
Body 3	135.65	1.0594e-003	0.16174	1.2794	2.3072	4937	2921	Solid	Solid	3221225475	22	77
Body 4	158.29	1.2363e-003	-12.692	0.78529	9.5652	3929	2345	Solid	Solid	3221225476	22	102
Body 5	158.42	1.2373e-003	-13.301	1.2968	1.5944	5764	3400	Solid	Solid	3221225477	22	120
Body 6	20.087	1.5688e-004	-9.2371	1.3747	-6.5276	2193	1237	Solid	Solid	3221225478	22	152

Figure 5-14: Selection information table.