

Ansys nCode DesignLife
新技術線上研討會
(高溫疲勞和潛變分析應用)

CAD MEN

虎門科技股份有限公司

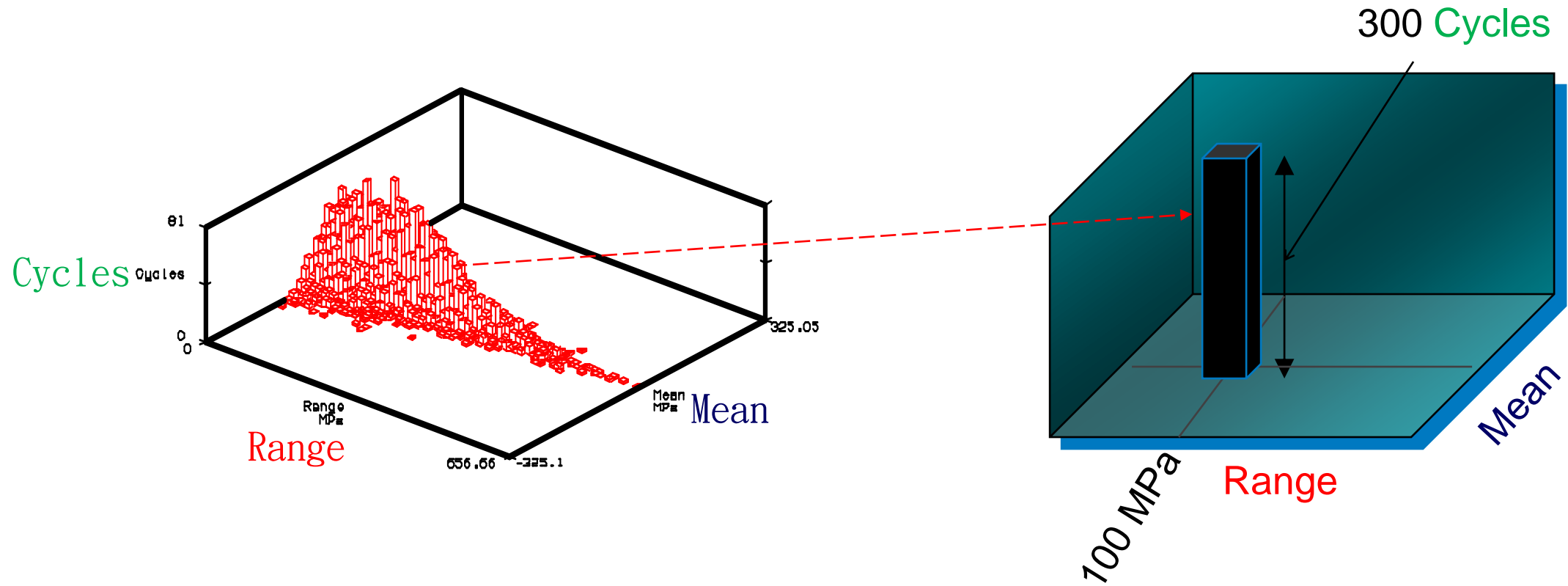
CAE事業部 鄒明嘉 (Warren)

Ansys

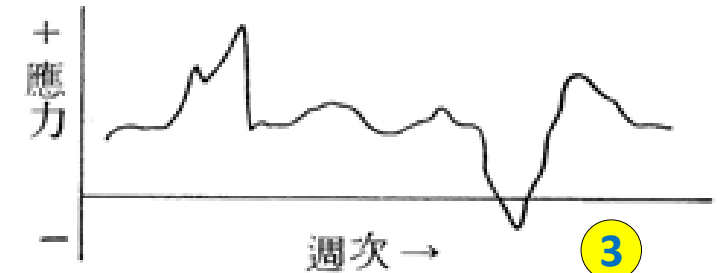
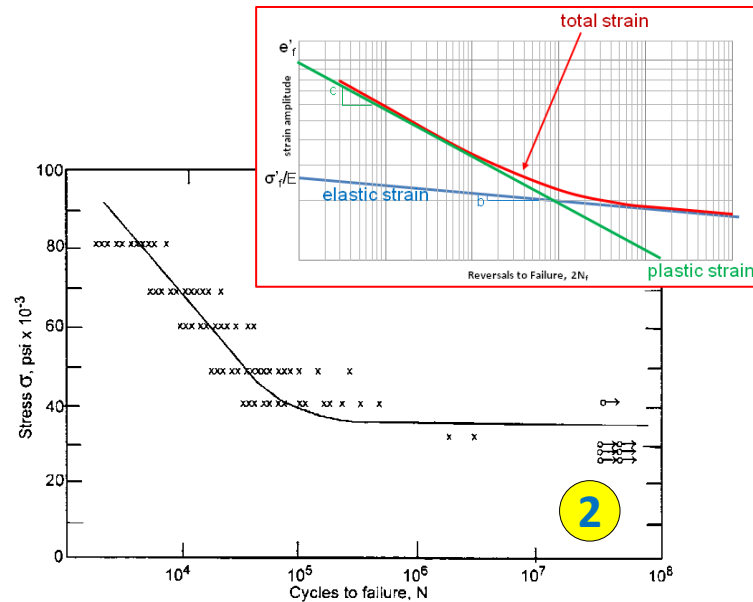
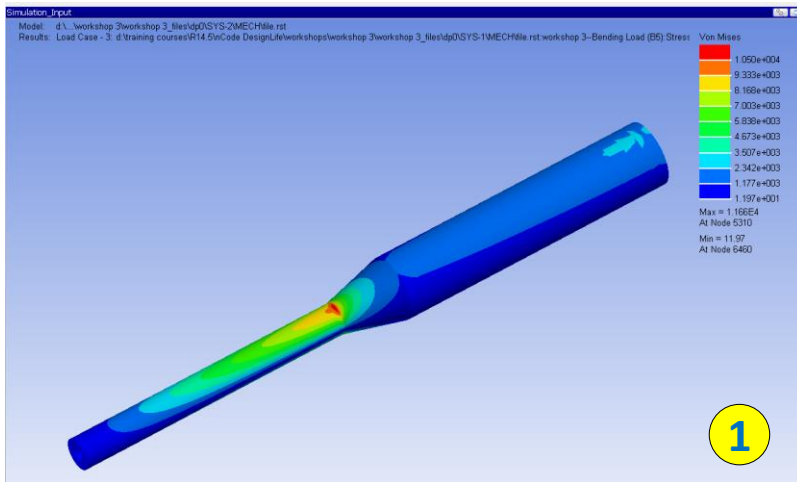
內容大綱：

1. 前言 nCode Highlight
2. 溫度相依的疲勞 Temperature dependent fatigue
3. 潛變斷裂 Creep rupture
4. 高溫結構疲勞分析 Thermo-mechanical fatigue

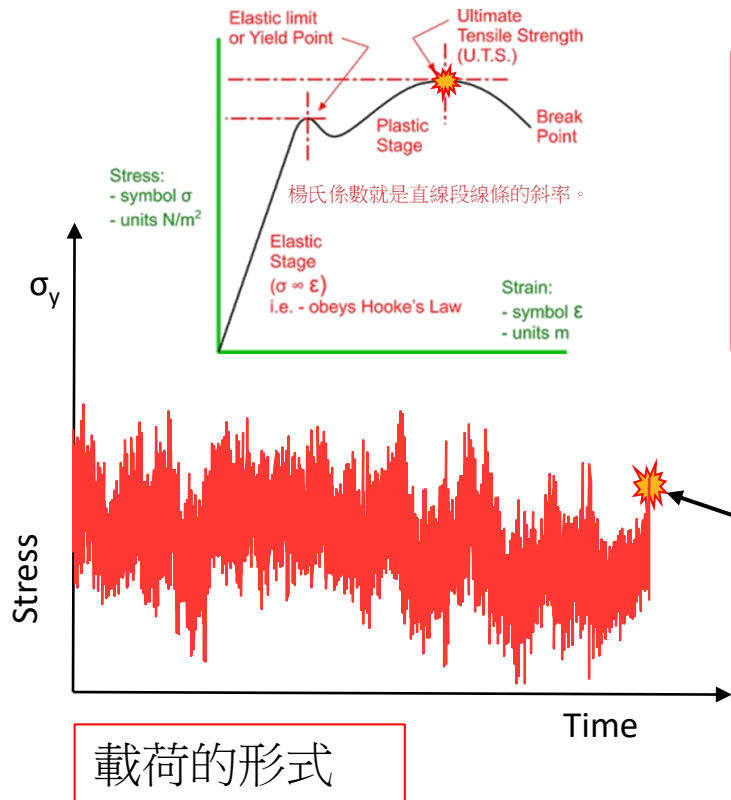
- 影響疲勞壽命的主要因素：次數(Cycles)、應力幅(Range)、平均應力(Mean)



1. 應力結果_ANSYS 應力分析
2. 疲勞曲線圖 → S-N 曲線 (應力疲勞曲線)、E-N 曲線(應變疲勞曲線)
3. 載荷的形式



- 疲勞曲線圖：S-N 曲線 (應力疲勞曲線)、E-N 曲線(應變疲勞曲線)



疲勞損傷是在波動載荷下裂紋的萌生 和/或 擴展

- 漸進性損傷
- 零件在多次負載作用後，似乎失去強度並變得疲倦，因此得名“疲勞”
- 幾乎所有的結構部件都受到循環加載

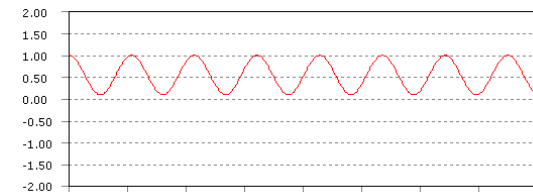
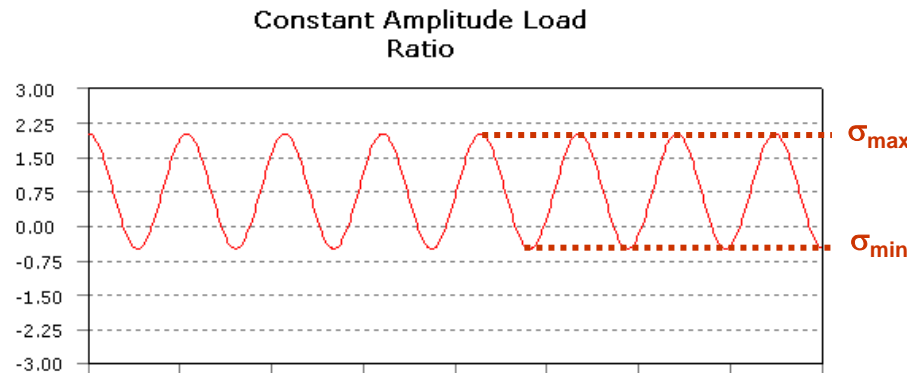
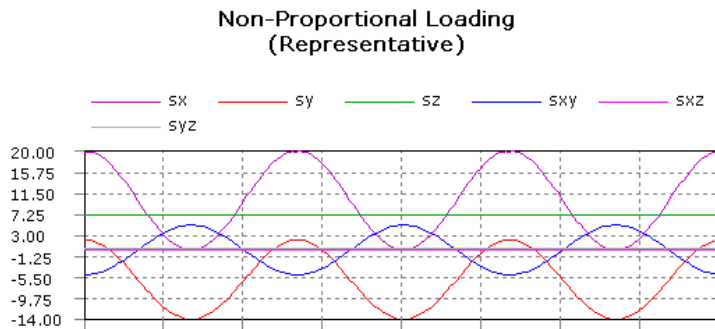
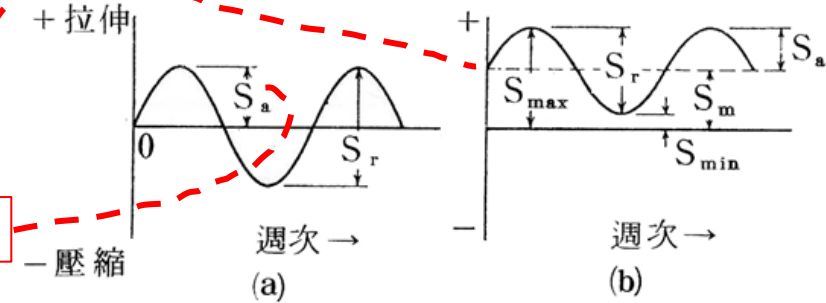
即使應力很低，在重複加載後也會發生疲勞破壞

恆定幅度(固定的振幅)、比例載荷、最小和最大的應力值

Consider the case of constant amplitude, proportional loading, with min and max stress values

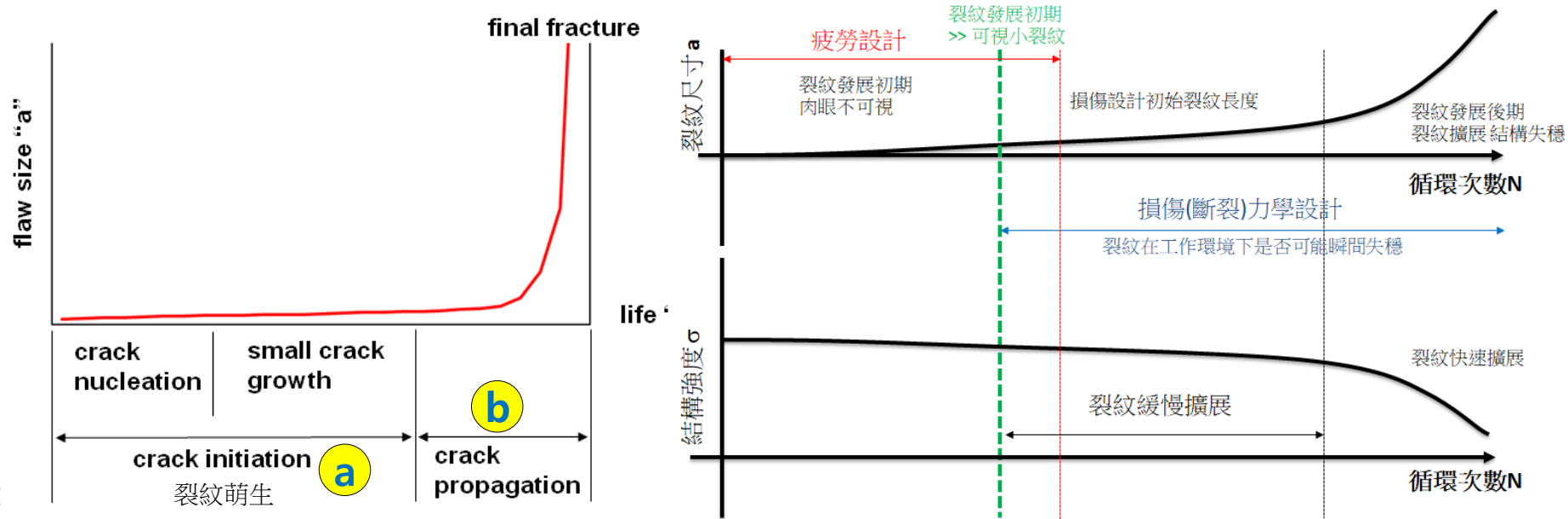
σ_{\min} and σ_{\max} :

- The **stress range** $\Delta\sigma$ is defined as $(\sigma_{\max} - \sigma_{\min})$ 應力範圍
- The **mean stress** σ_m is defined as $(\sigma_{\max} + \sigma_{\min})/2$ 平均應力
- The **stress amplitude** or **alternating stress** σ_a is $\Delta\sigma/2$ 應力幅
- The **stress ratio** R is $\sigma_{\min}/\sigma_{\max}$ 應力比
- **Fully-reversed loading** occurs when an equal and opposite load is applied. This is a case of $\sigma_m = 0$ and $R = -1$.
- **Zero-based loading** occurs when a load is applied and removed. This is a case of $\sigma_m = \sigma_{\max}/2$ and $R = 0$.



疲勞發展過程

• 疲勞失效



疲勞是一個發展的過程：

從 1.裂紋萌生到 2.裂紋擴展以至 3.最後斷裂。

裂紋失穩擴展斷裂是一個很快的過程，對疲勞壽命影響很小，在疲勞分析中一般不考慮。

一般只考慮裂紋萌生及裂紋擴展。

- a. 進行裂紋萌生壽命分析，一般採用應變疲勞分析(局部塑性應變即裂紋形成，剛度改變，應變也隨之變化)
- b. 進行裂紋擴展壽命分析，一般採用斷裂力學的方法

材料不會產生宏觀的塑性變形，一般直接採用應力疲勞分析。

Abs Max Principal

Signed VonMises

破壞的定義：裂紋萌生 → 1)脆性材料(斷裂) 2)延性材料(可視小裂紋)

裂紋萌生擴展至斷裂時間很短

裂紋萌生後，有相當長的一段擴展階段，不應當計入裂紋萌生壽命



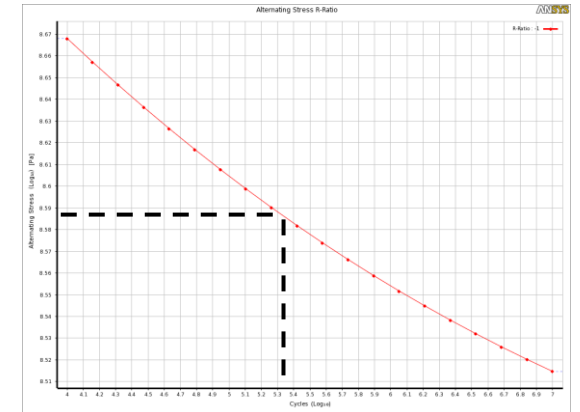
➤ 應力壽命 (SN) 使用彈性計算的應力和應力與循環疲勞失效曲線 (S-N 曲線)

-- 假設應力會導致疲勞 (假設疲勞損傷是由波動應力產生的)

-- 僅適用於高週疲 (僅限於低應力/高循環應用)

壽命高於 1~10 萬次以上循環

延展性金屬大於 10 萬次循環



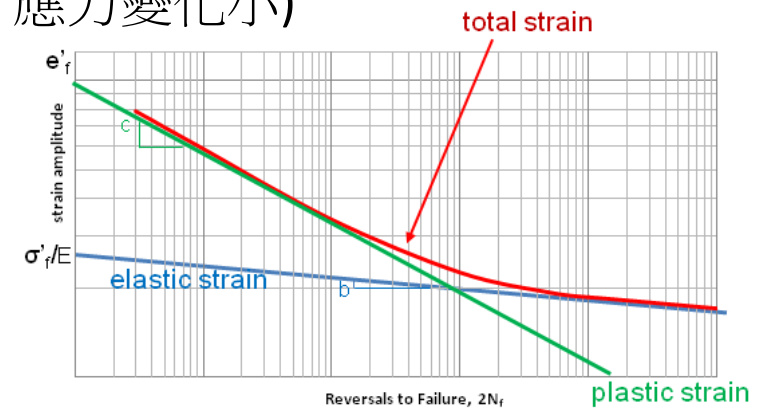
➤ 應變壽命 (EN) 使用彈-塑性應變和應變壽命關係方程

-- 假設局部塑性應變會導致疲勞 (延性好的材料降伏後，應變變化大、應力變化小)

-- 適用於低週疲勞和高週疲勞 (載荷水平超過降伏應力)

-- 預測裂紋萌生

壽命低於 1 萬次循環

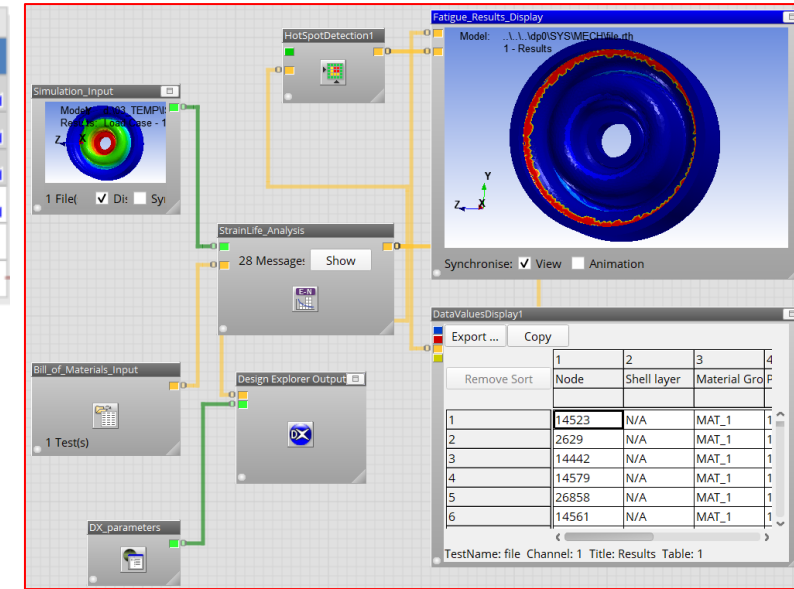
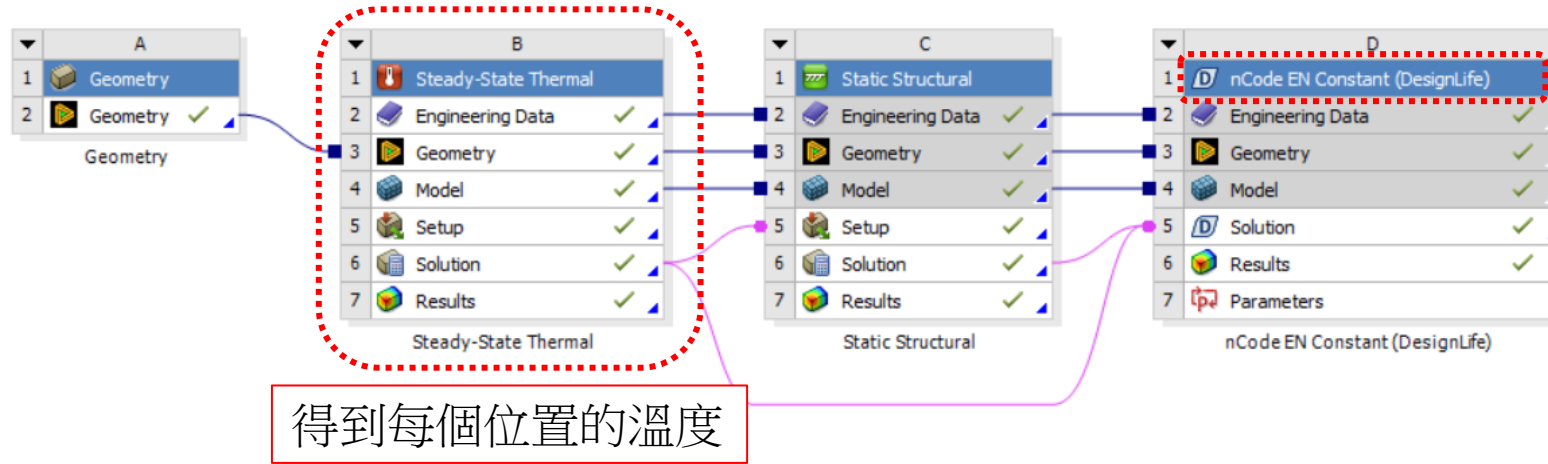


內容大綱：

1. 前言 nCode Highlight
2. 溫度相依的疲勞 Temperature dependent fatigue
3. 潛變斷裂 Creep rupture
4. 高溫結構疲勞分析 Thermo-mechanical fatigue

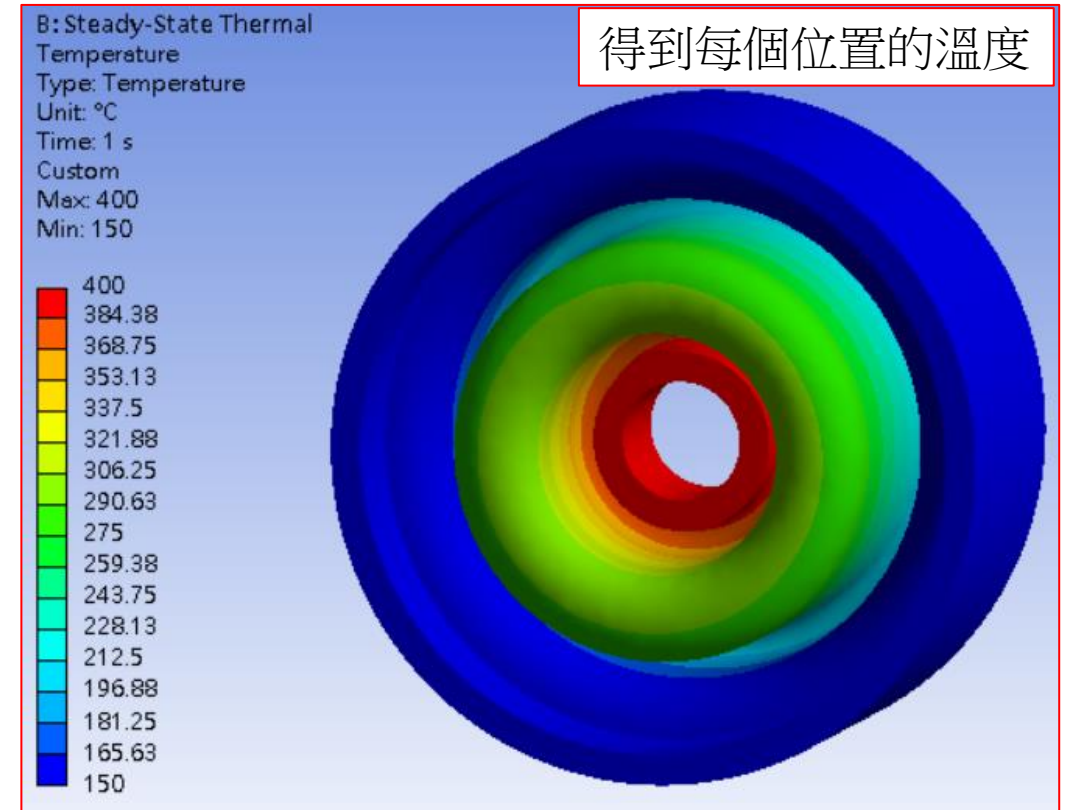
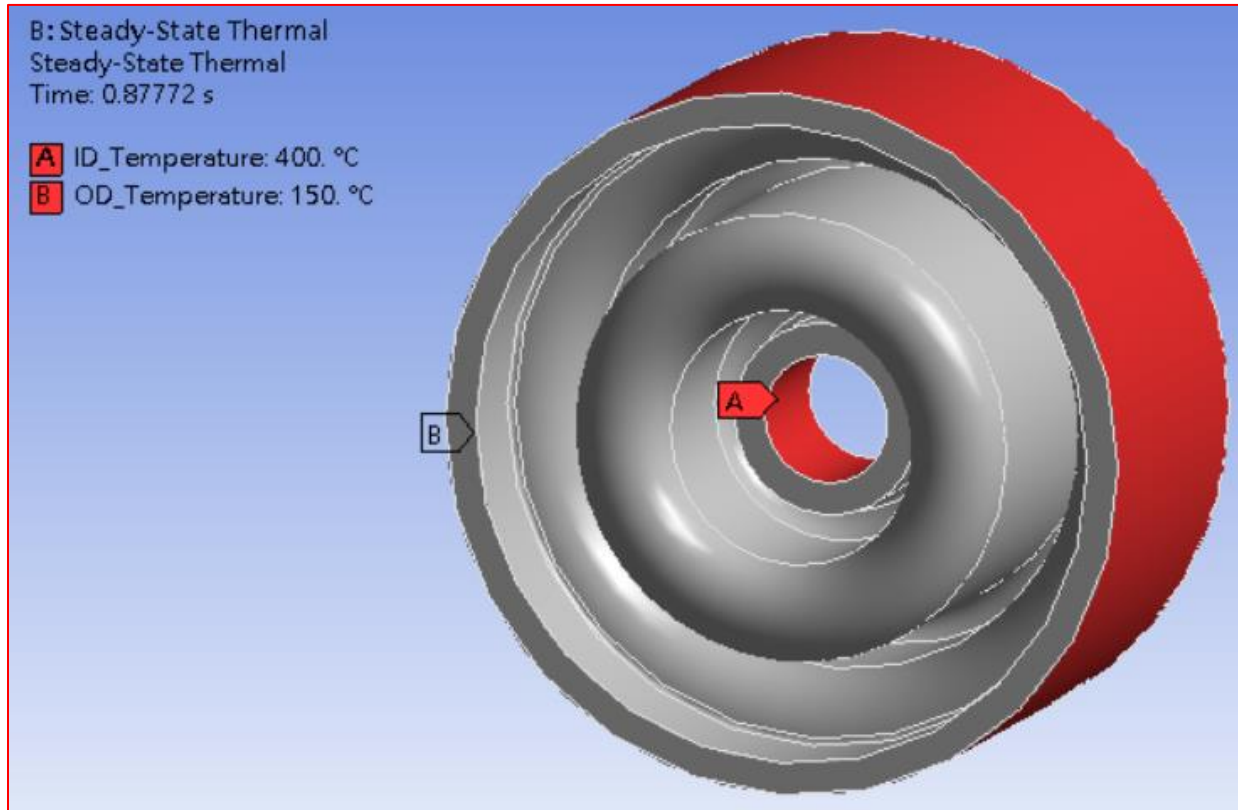
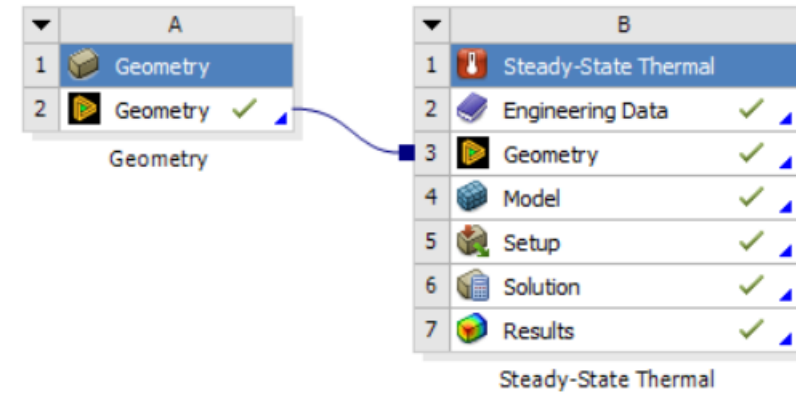
溫度相依的疲勞分析流程

• 工作流程

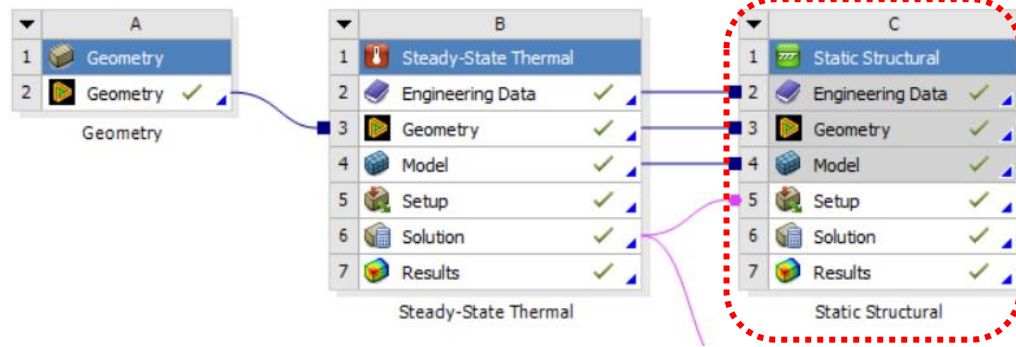


Isothermal Fatigue Analysis (等溫疲勞)

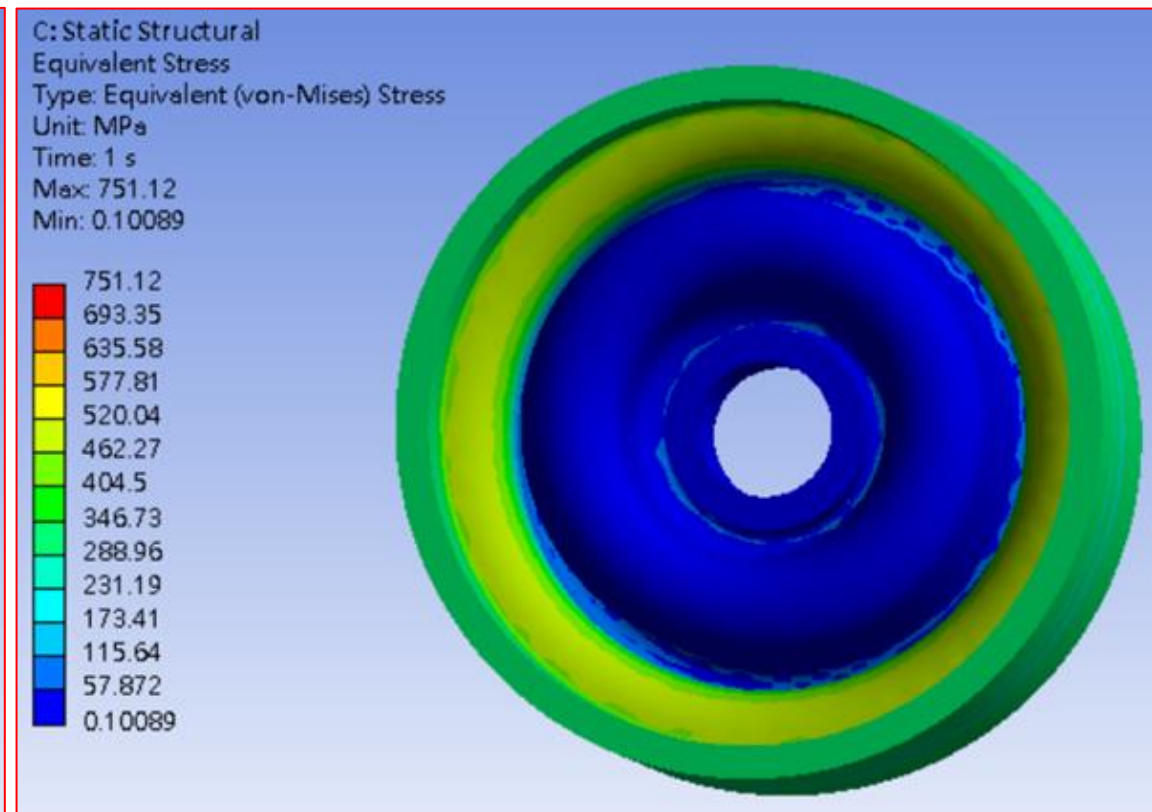
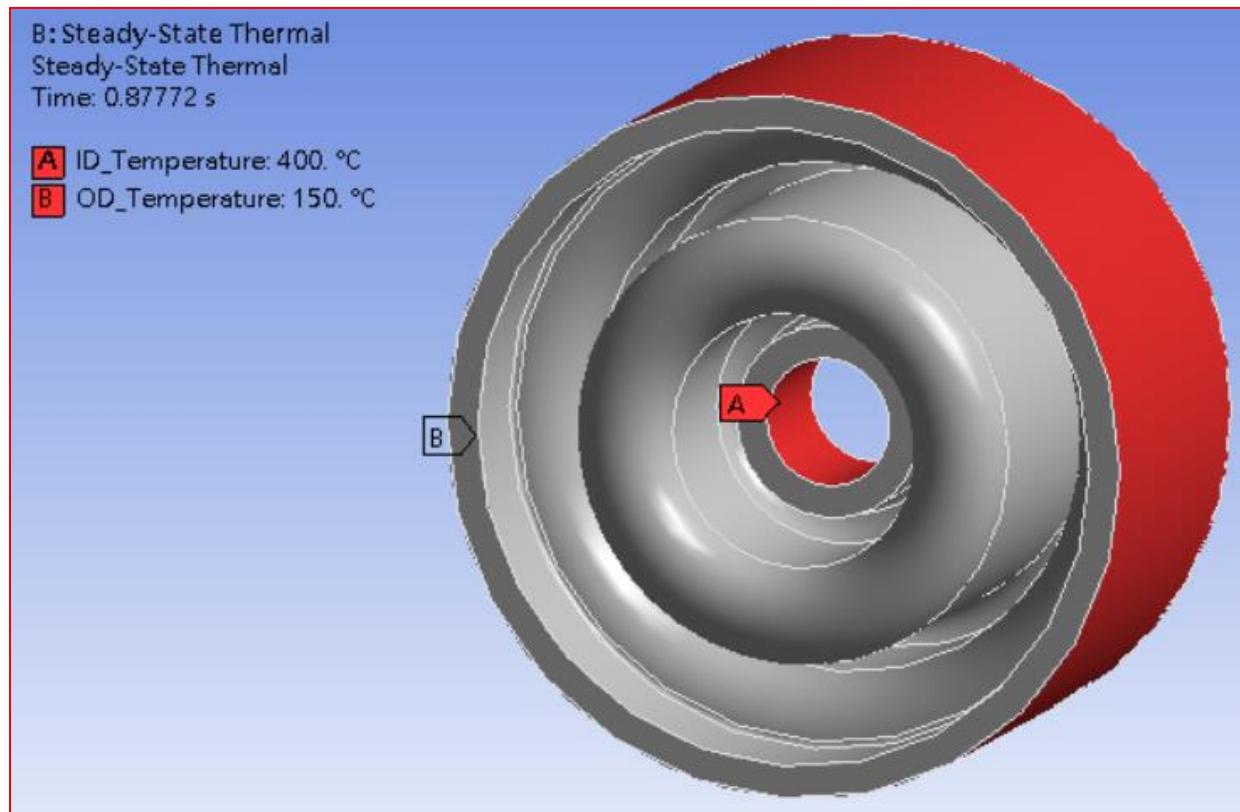
- 穩態熱傳



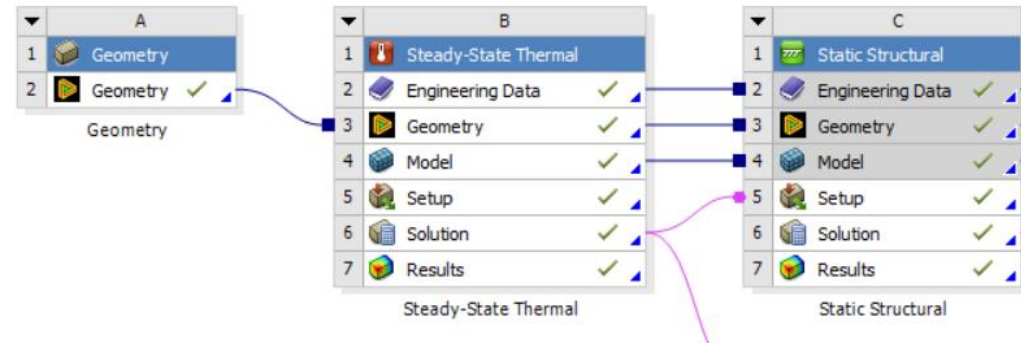
Isothermal Fatigue Analysis (等溫疲勞)



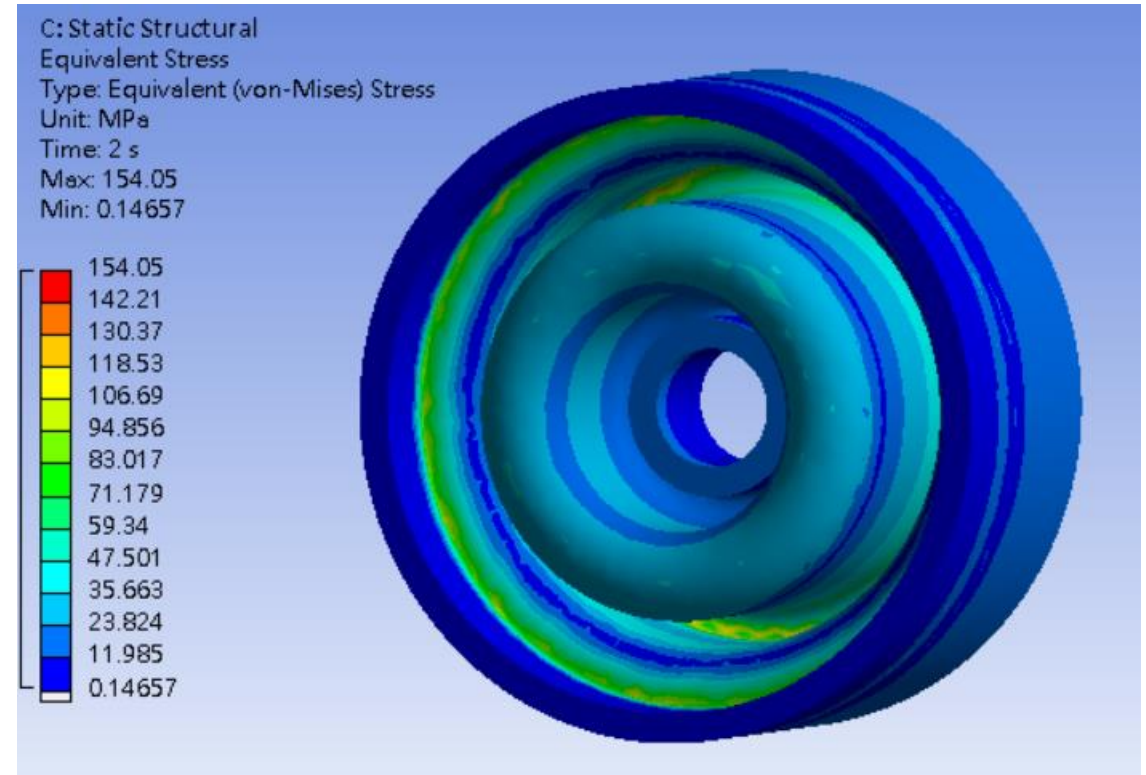
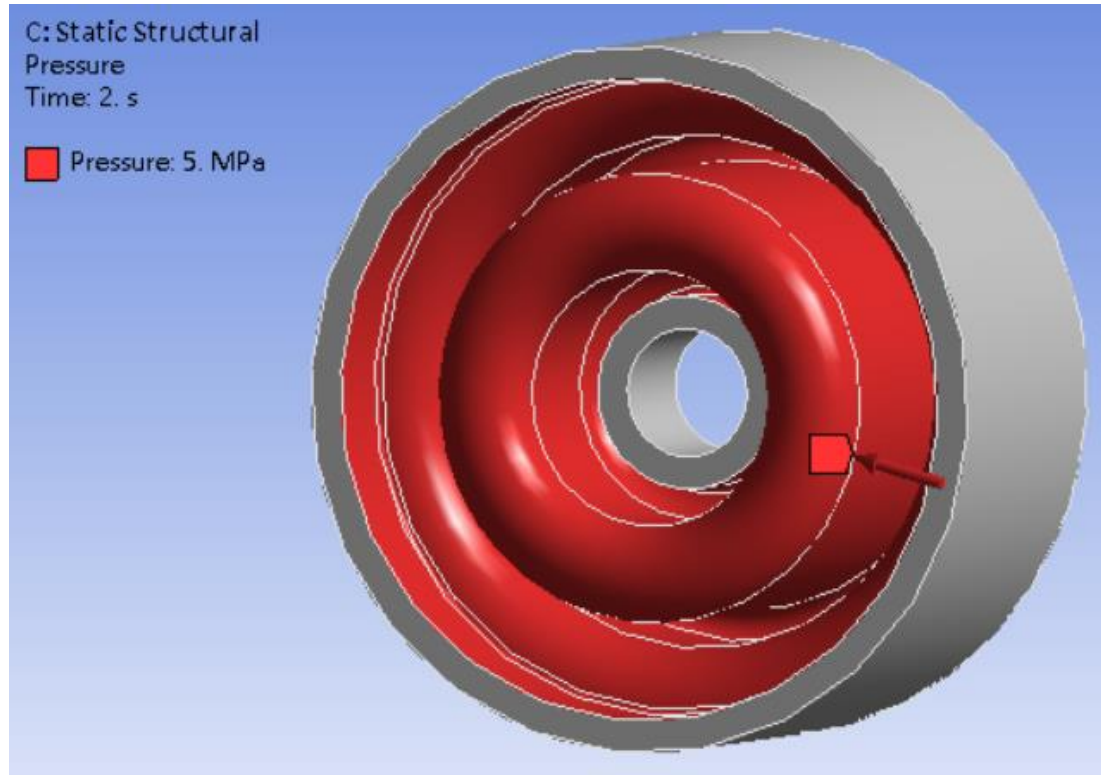
- 熱漲冷縮 → 熱變形 → 熱應力



Isothermal Fatigue Analysis (等溫疲勞)

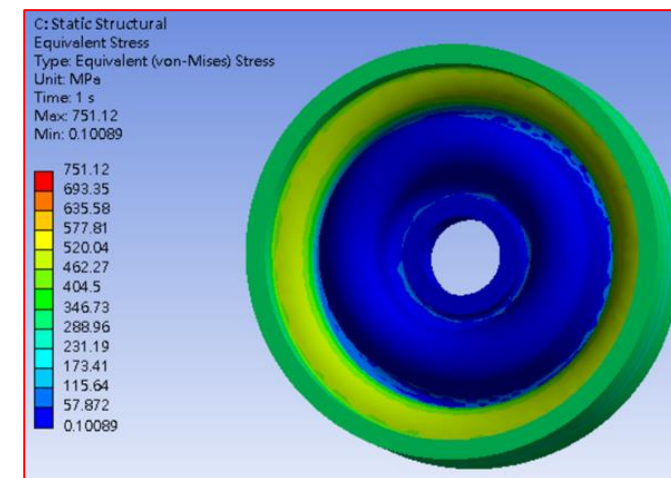
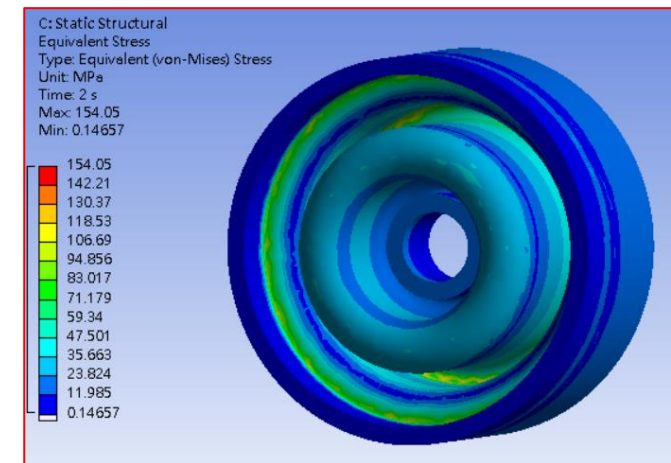
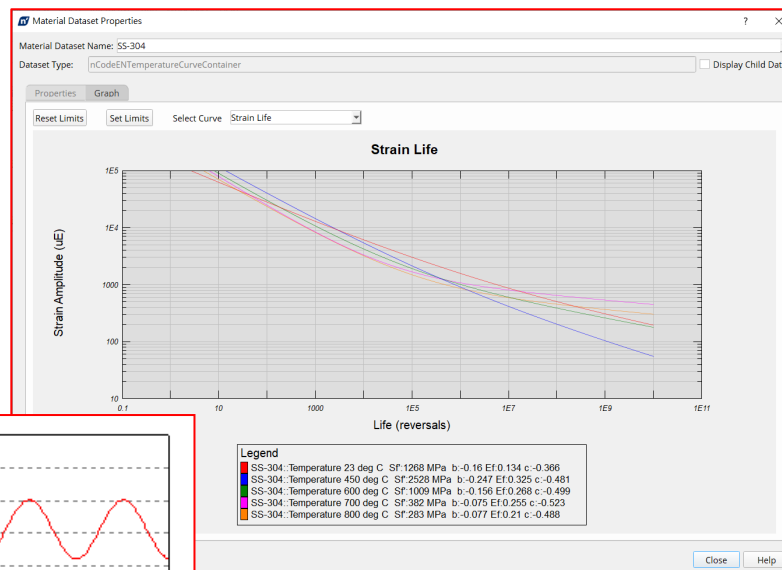
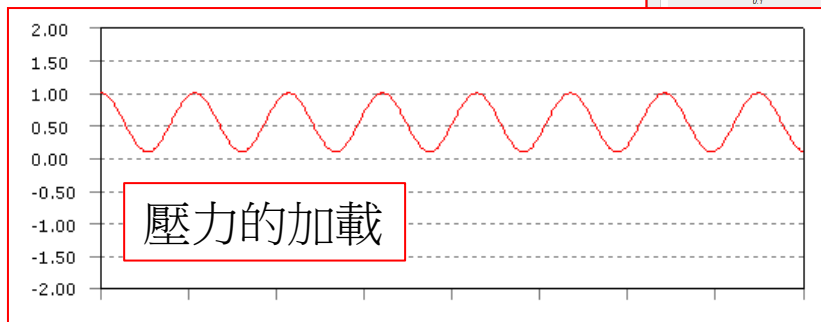


- 施加壓力的應力結果

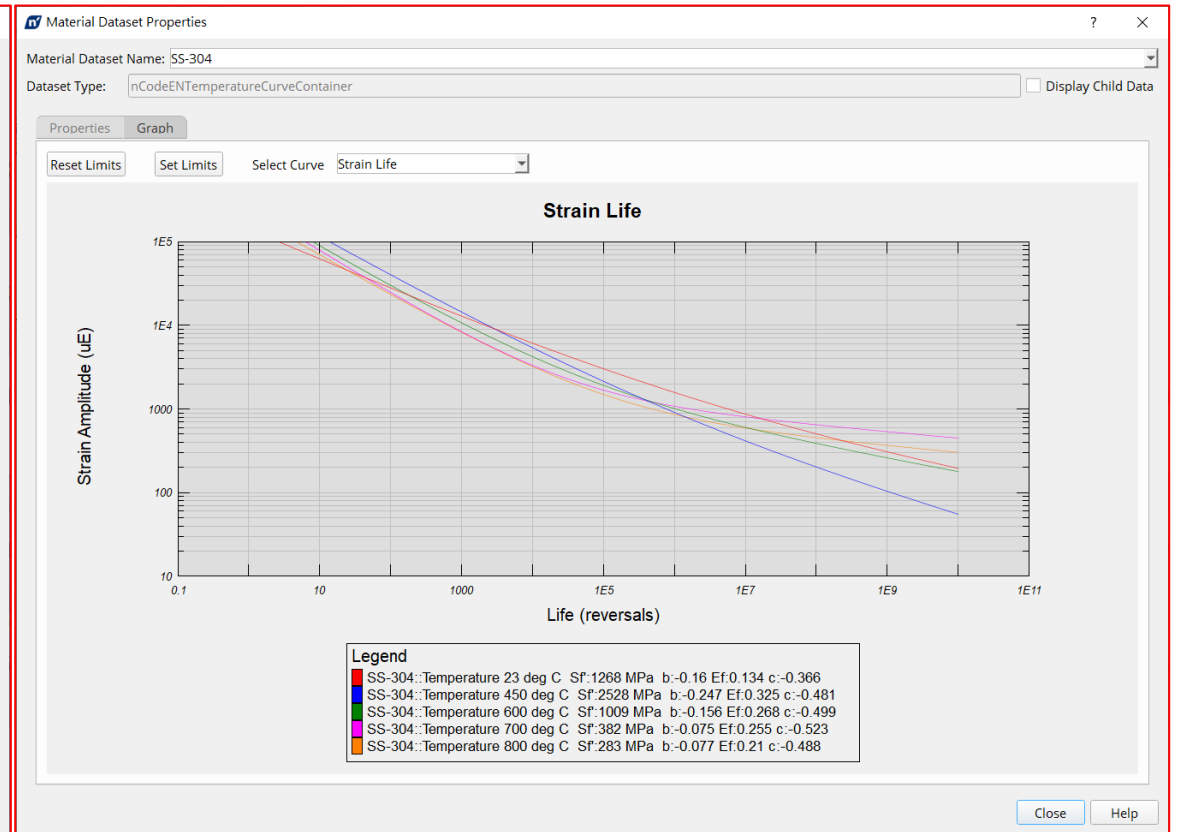


1. 應力結果_ANSYS 應力分析，包含熱應力
2. 疲勞曲線圖 → E-N 曲線(溫度相依的應變疲勞曲線)
3. 載荷的形式

等溫，無循環的熱應力

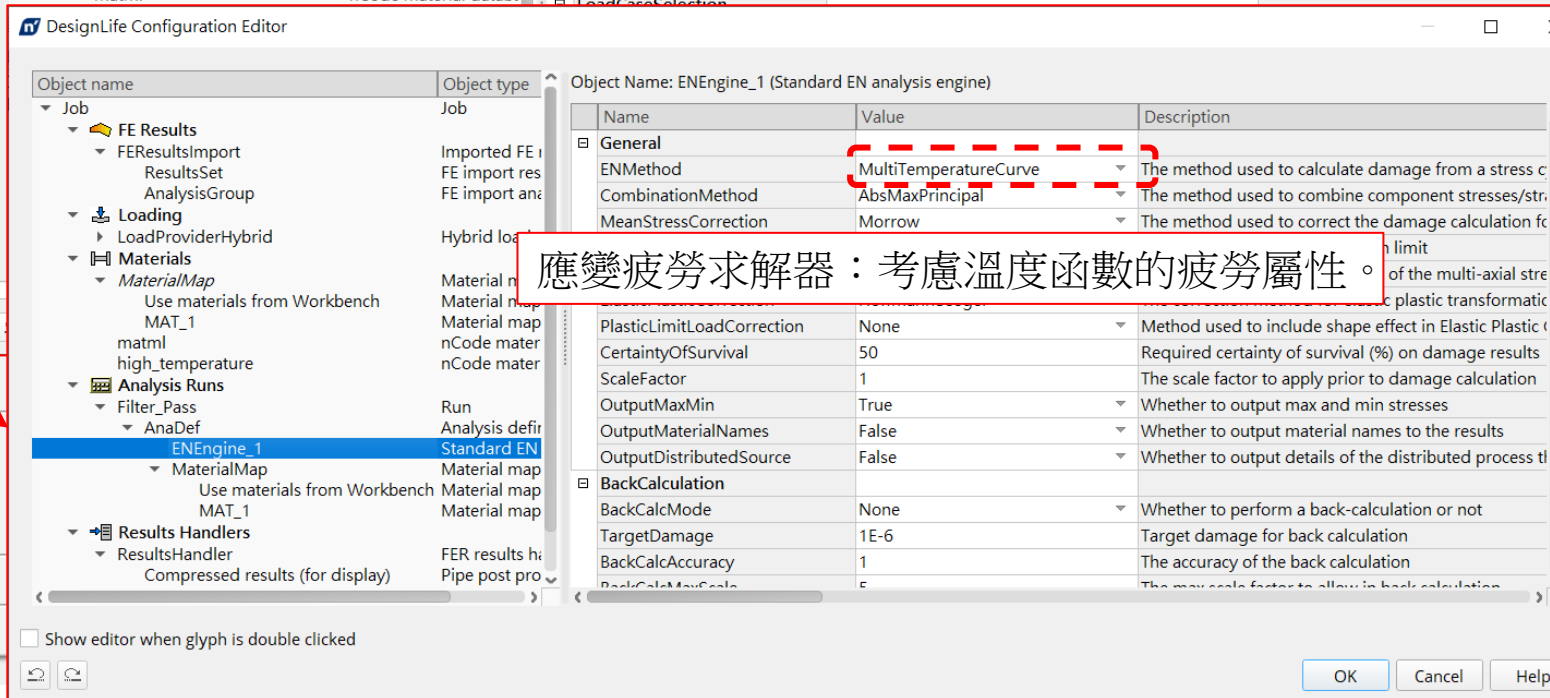
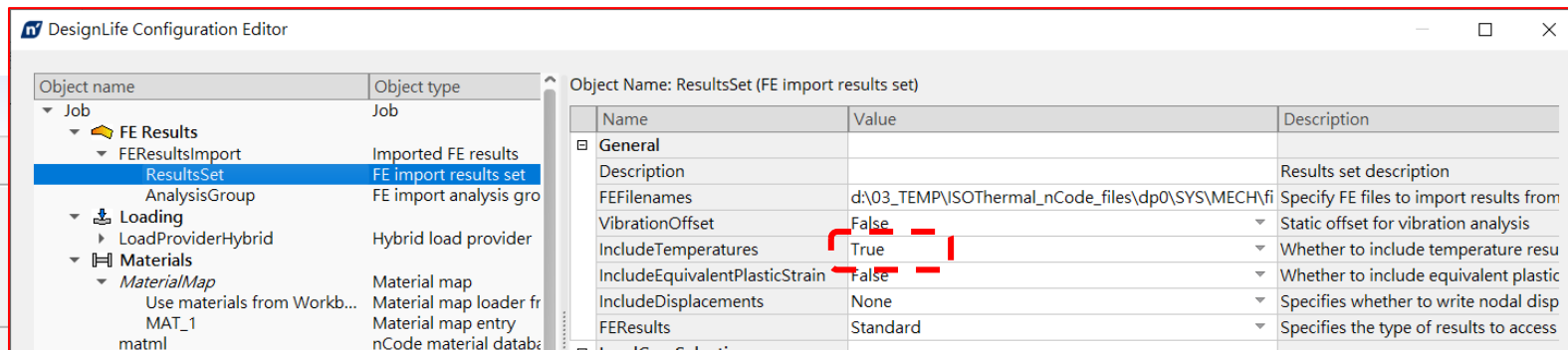
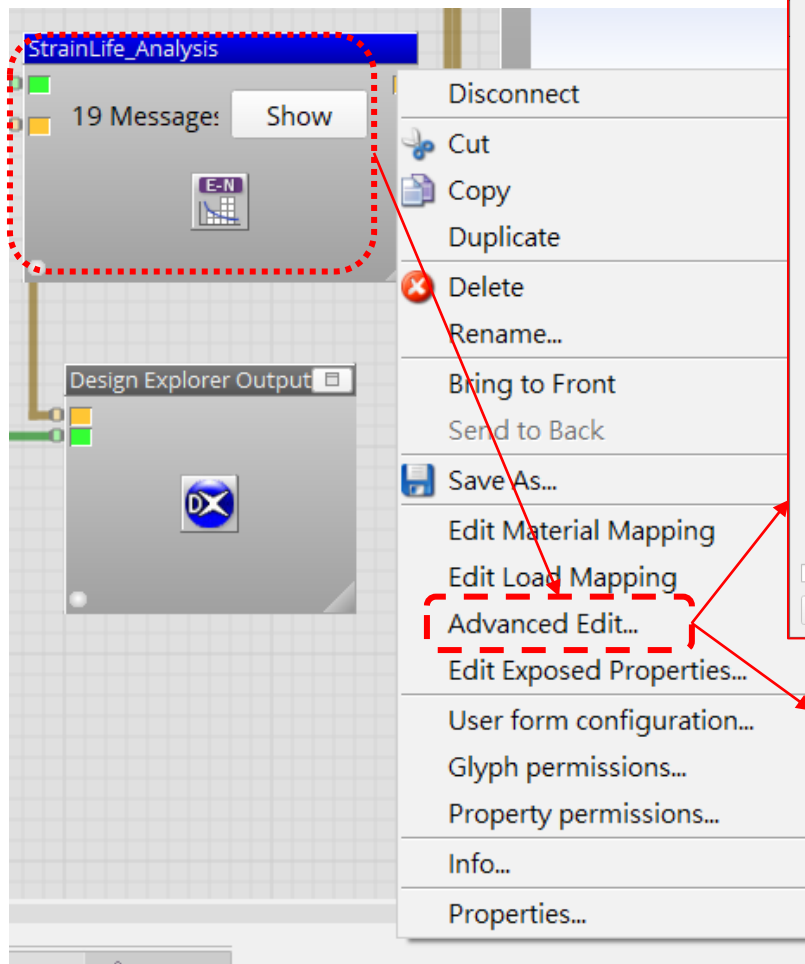


- 疲勞曲線圖 → E-N 曲線_在不同溫度下的疲勞曲線



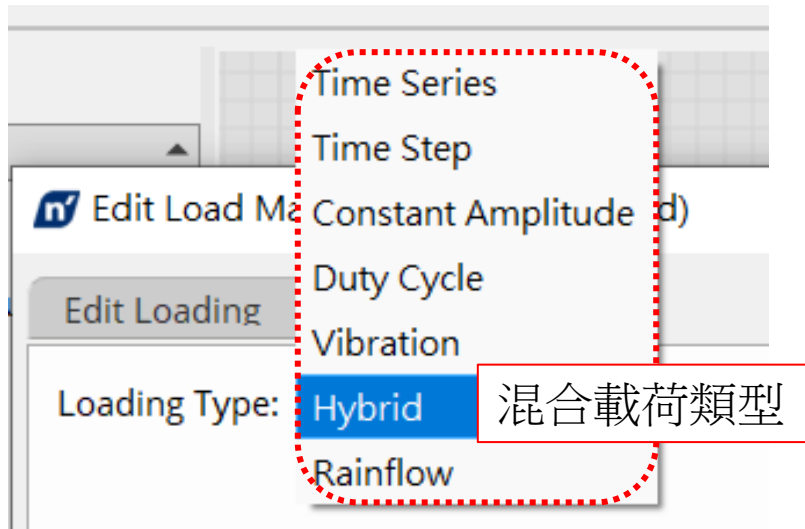
設定溫度選項

從 ANSYS Steady-State Thermal
導入溫度



Loading Type (載荷類型)

- *.rth file (熱傳結果檔)



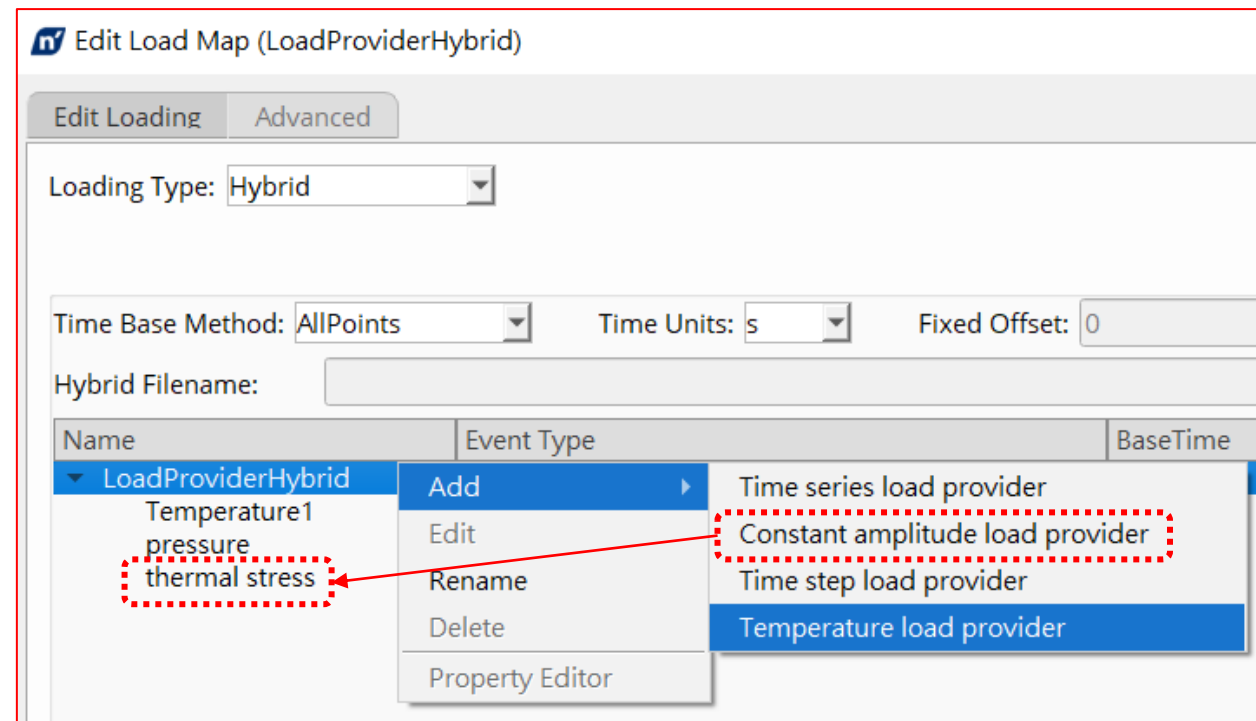
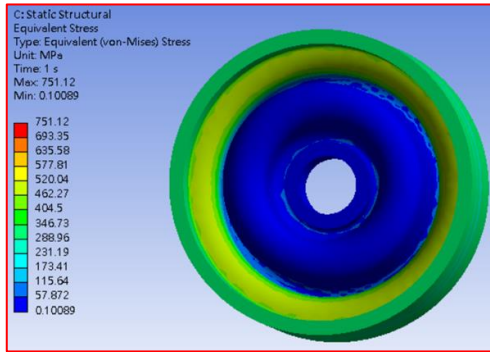
溫度載荷可提供多個溫度狀態。可以指定溫度的最大值、平均值或最小值，也可以設置為特定步長。此例中，只有一組解。

執行：溫度相依的疲勞分析

Name	Event Type	BaseTime
LoadProviderHybrid		
Temperature1		
pressure		
thermal stress		

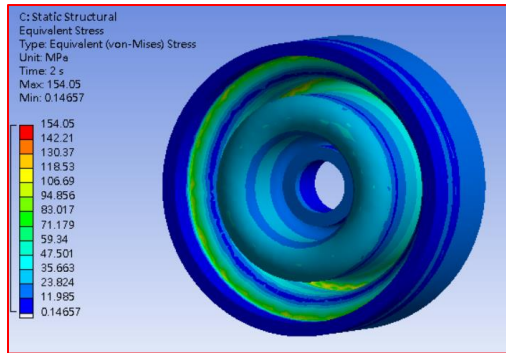
載荷工況(1)

- *.rst file 熱應力結果檔



載荷工況(2)

- *.rst file 受壓力時的應力結果結果檔



Edit Load Map (LoadProviderHybrid)

Edit Loading | Advanced

Loading Type: Hybrid

Time Base Method: AllPoints | Time Units: s | Fixed Offset: 0

Hybrid Filename:

Name	Event Type	BaseTime
LoadProviderHybrid		
Temperature1		
pressure		
thermal stress		

Context Menu for LoadProviderHybrid:

- Add (selected)
- Edit
- Rename
- Delete
- Property Editor

Options for Add:

- Time series load provider
- Constant amplitude load provider (highlighted)
- Time step load provider
- Temperature load provider

Edit Load Map (pressure)

Edit Loading | Advanced

Loading Type: Constant Amplitude

Load Case Descriptions: All

Available FE Load Cases

Description
1 - d:\03_TEMP\ISOThermal_nCode_files\d...
2 - d:\03_TEMP\ISOThermal_nCode_files\d...

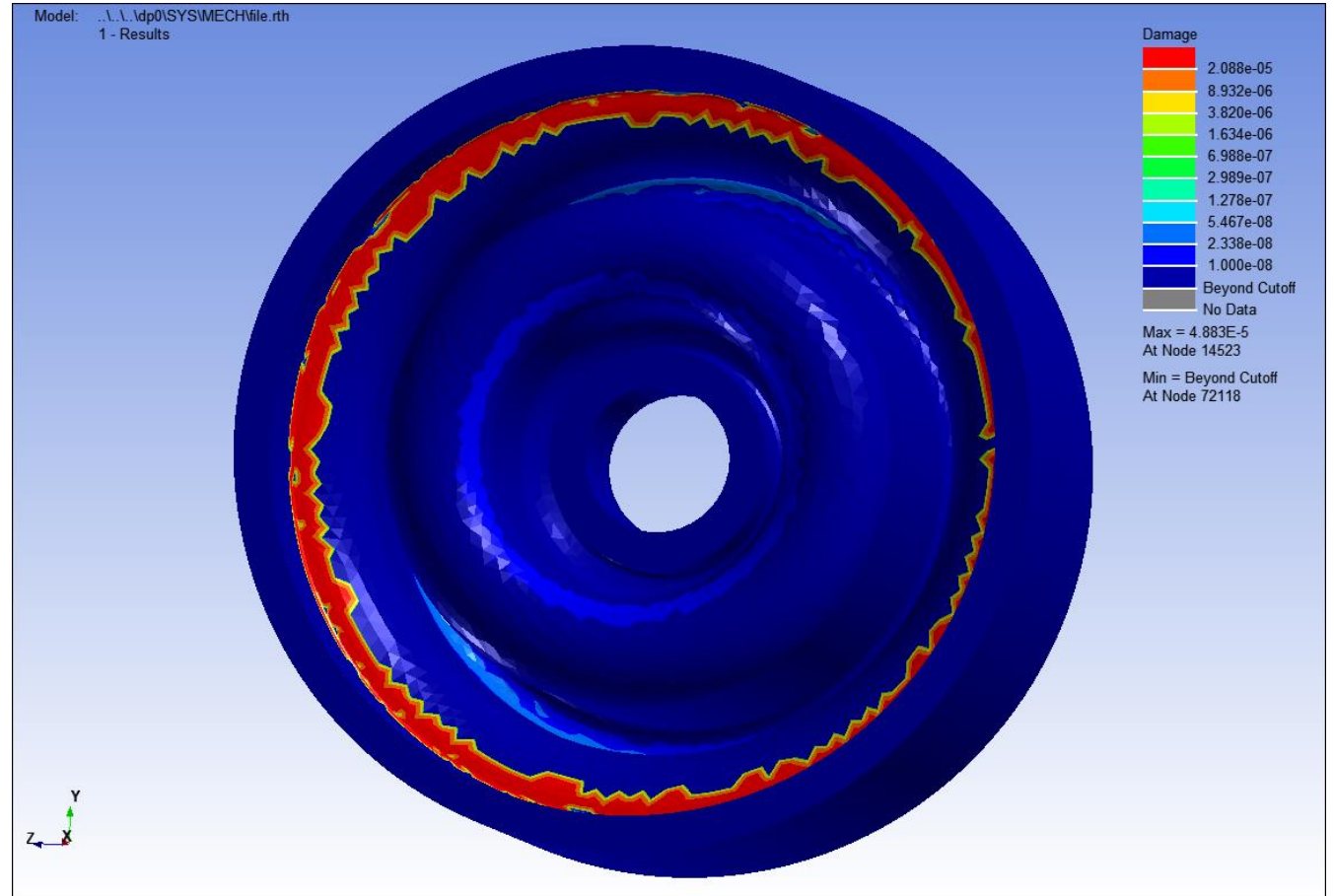
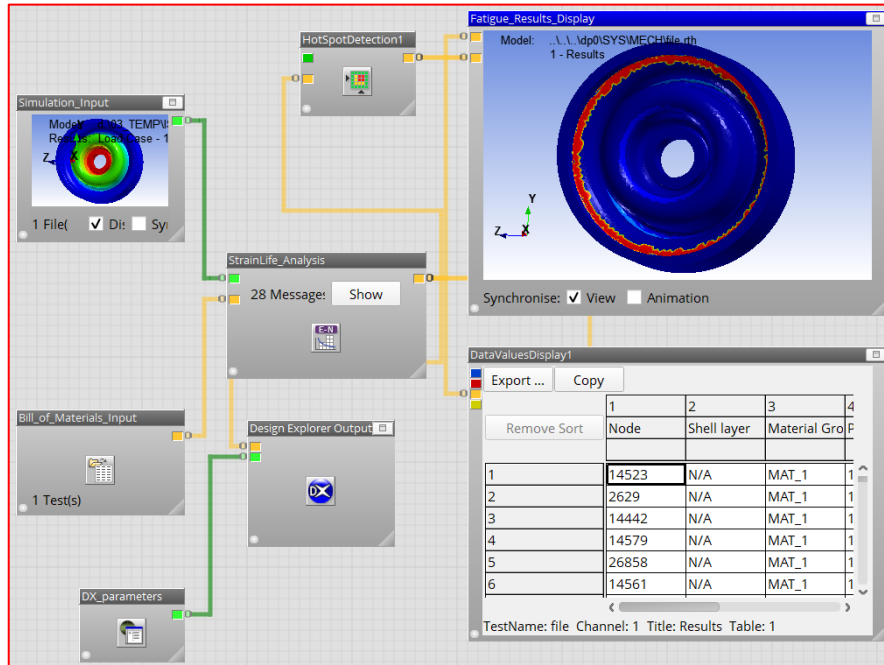
Load Case Assignments

Load Case	Description	Max Factor	Min Factor
1	2 - d:\03_TEMP\ISOThermal_nCode_files\d...	1	0

Graph: Sinusoidal wave between -2.00 and 2.00.

Buttons: OK, Cancel, Help

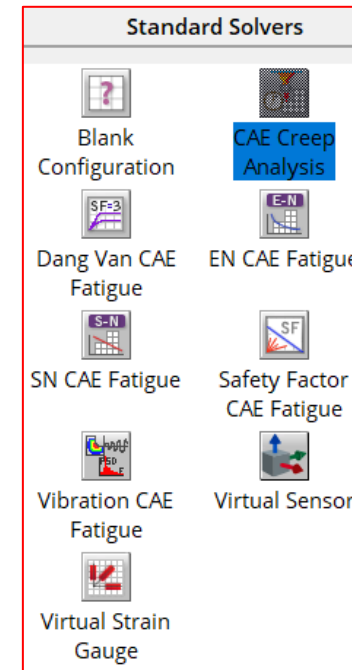
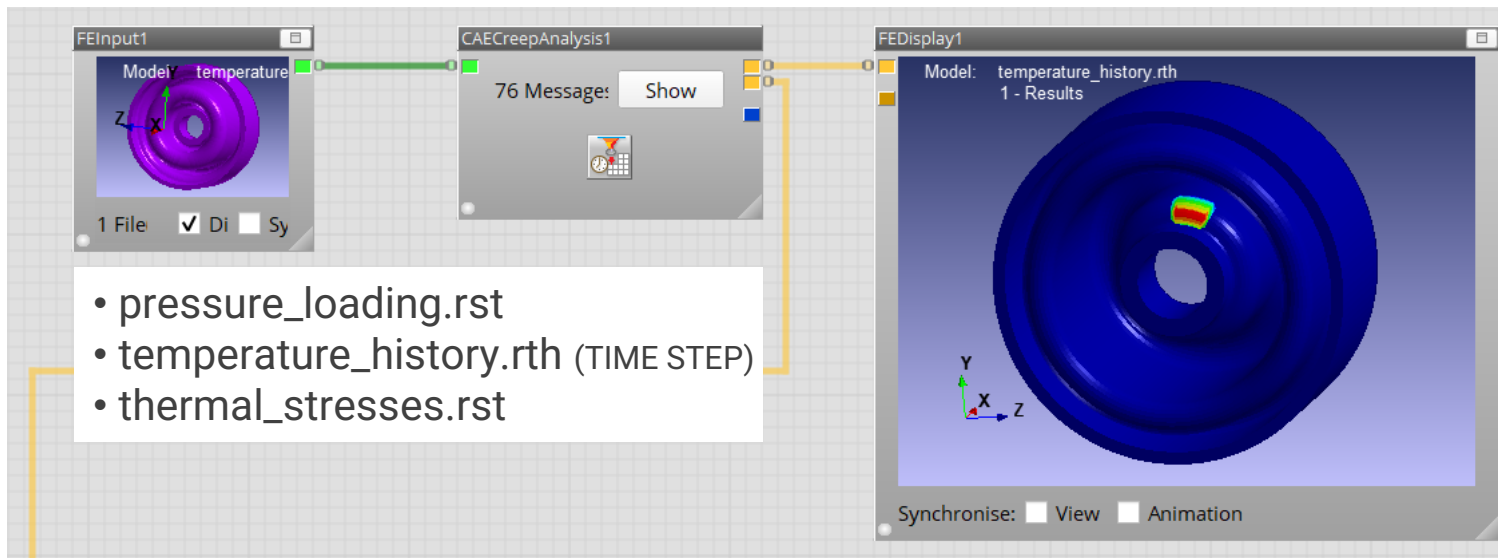
• 工作流程



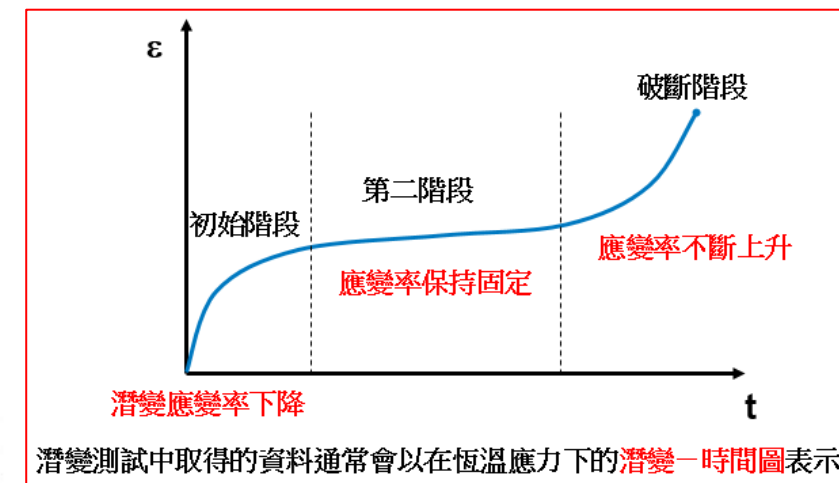
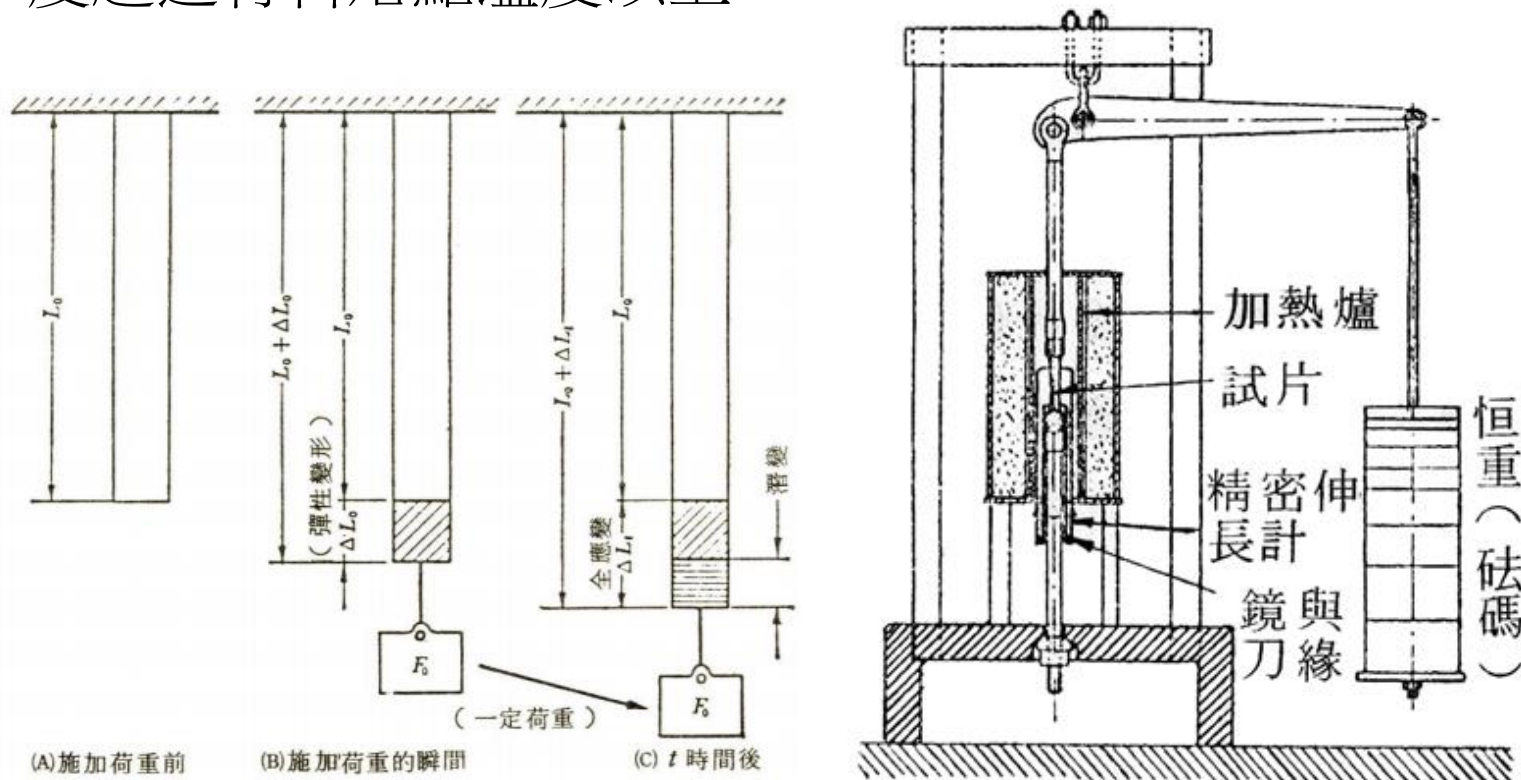
內容大綱：

1. 前言 nCode Highlight
2. 溫度相依的疲勞 Temperature dependent fatigue
3. 潛變斷裂 Creep rupture
4. 高溫結構疲勞分析 Thermo-mechanical fatigue

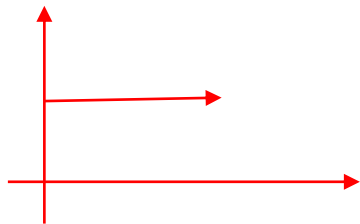
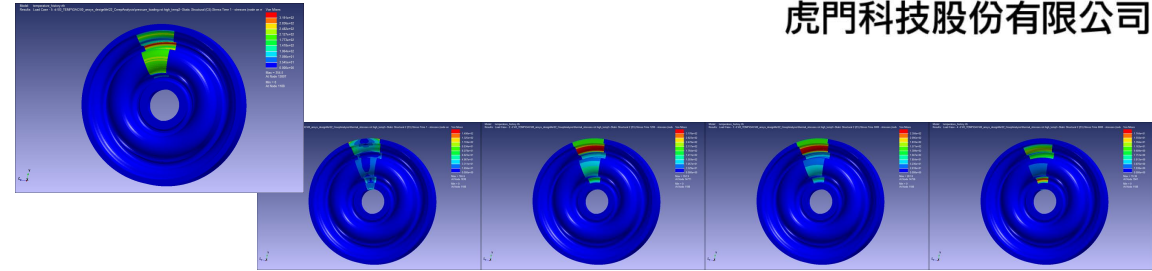
- 當受力元件的溫度足夠高時(通常大於材料熔點Kelvin的 30%)，然後開始發生潛變損傷，無論壓力是否循環，都會發生這種損傷。
- 足夠高溫度下的靜態應力最終會導致組件失效，如果應力也在變化，那麼也會發生疲勞損傷。
- 設計高溫元件時可能需要考慮潛變。



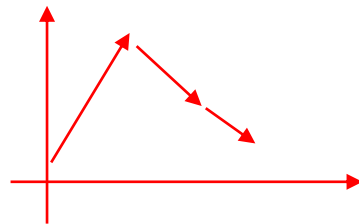
➤ 變形隨時間徐徐進行之現象稱之為潛變，通常這種現象發生在材料放在環境溫度超過材料熔點溫度以上。



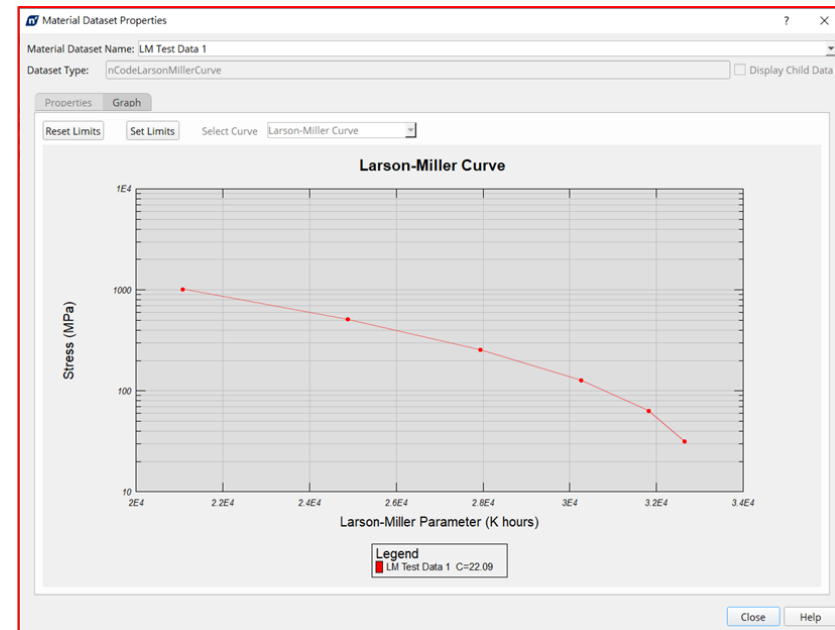
1. 應力結果_ANSYS 應力分析，包含熱應力
2. 潛變曲線圖 → Larson-Miller (拉森-米勒) 參數曲線
3. 載荷的形式



固定載荷

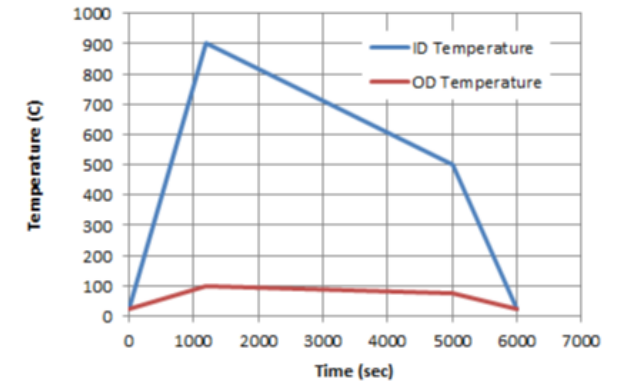
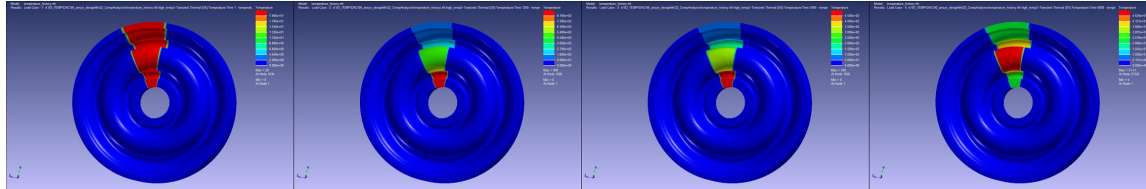


來自於軟體 TIME STEP 的加載

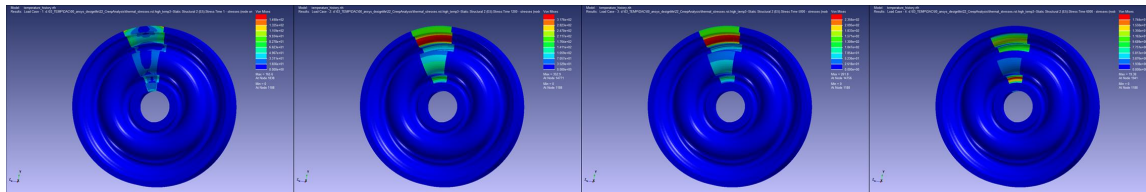


➤ 有限元分析分三個階段進行，產生三組 ANSYS 結果檔。

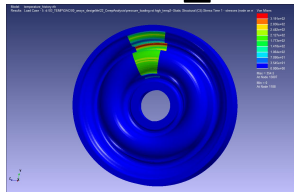
- Temperature_history.rth 包含時間 1、1200、5000 和 6000 秒的模型溫度。



- Thermal_stresses.rst 包含四組波紋管在時間 1、1200、5000 和 6000 秒處的應力，這是由於 temperature_history.rth 中描述的溫度歷程引起的熱膨脹。



- Pressure_loading.rst 包含由施加到波紋管一個面的 5 MPa 壓力載荷引起的單一應力狀態。



Larson-Miller

拉森-米勒參數

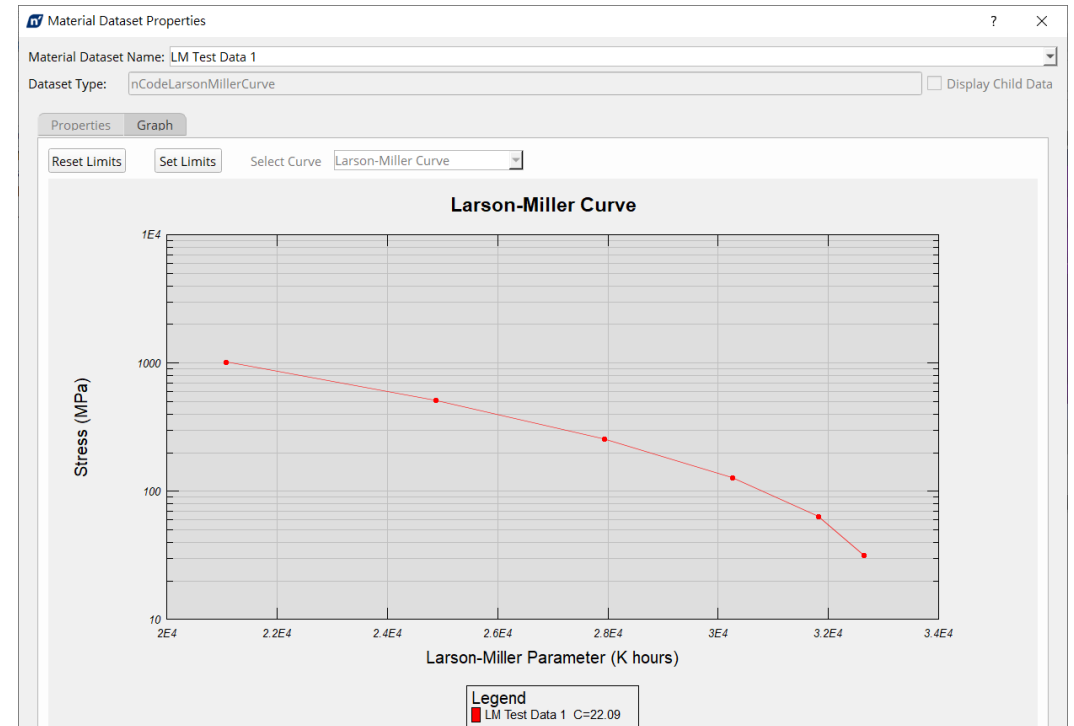
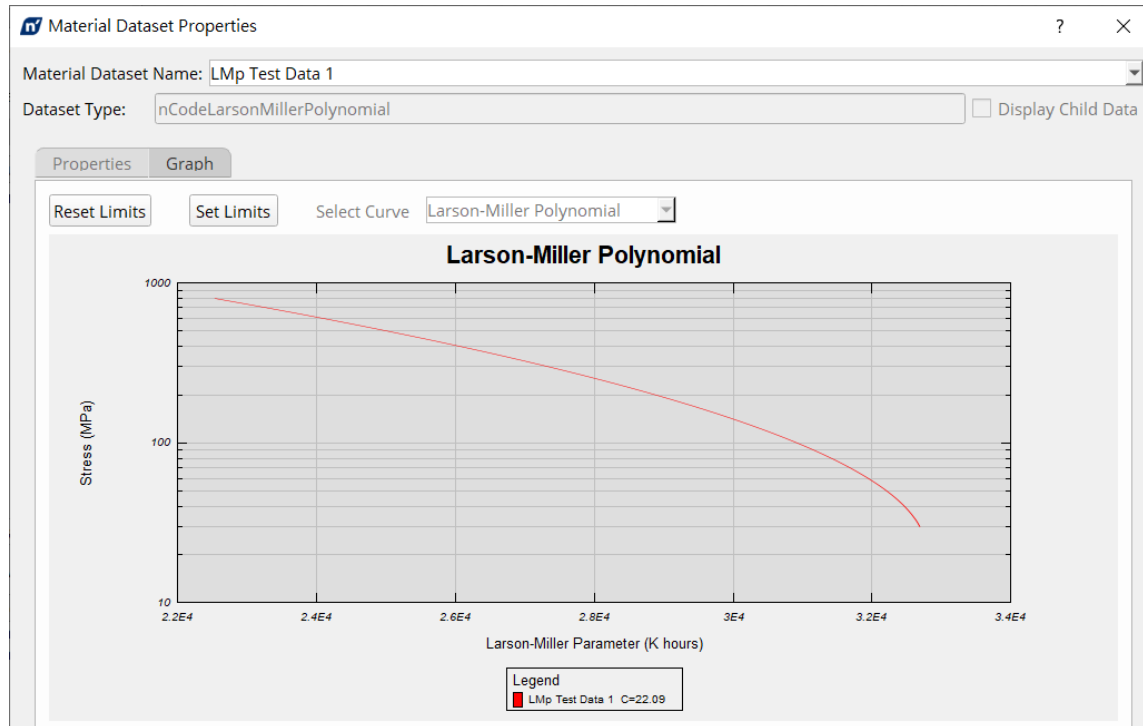
$$P(\sigma) = T \left(C + \frac{1}{2} \log \sigma + \log t_r \right) \times 10^{-3}$$

其中 C 是拉森-米勒係數，為常數；T 為試驗溫度 K；tr 為斷裂時間 h。

溫度 (°C)	試驗應力 (σ , / Mpa)	斷裂時間 (t_r / h)	P(σ)
950	120	1.3	25.87077
	92	5.0	26.5157
	78	10.1	26.8453
	62	31.0	27.37998
	49	76.5	27.79727

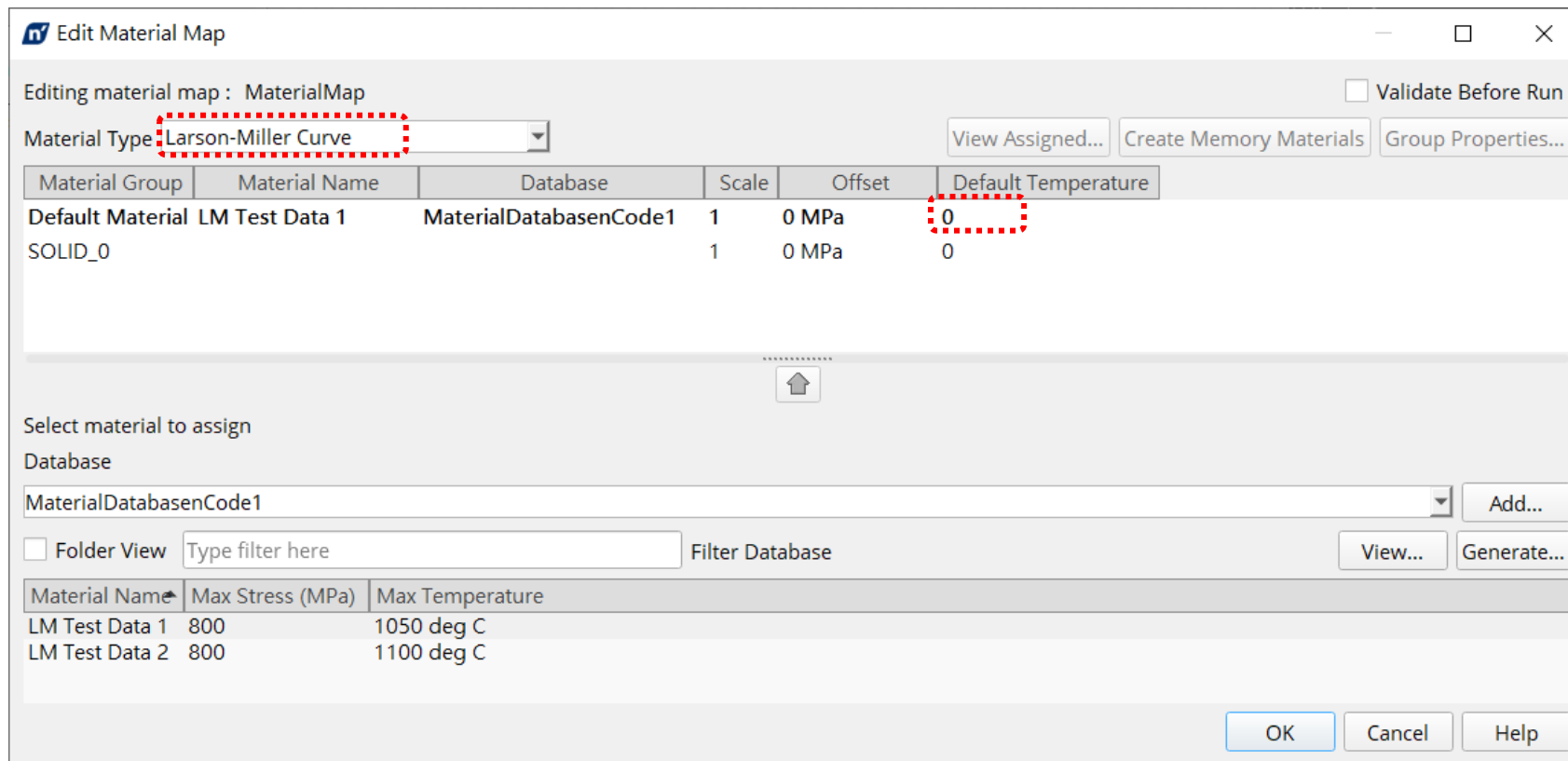
把試驗資料每一試驗點數據，即應力、溫度、時間組成的一組資料按式算出每個試驗應力 σ 下相應的 P(σ) 拉森-米勒參數值。

- 有兩種方法可以定義 Larson-Miller 材料，多項式擬合或輸入曲線點。

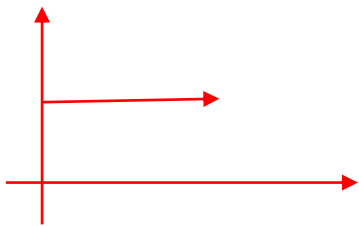


該曲線將應力與 Larson-Miller 參數相關聯，該參數被定義為壽命和溫度的函數。給定應力和溫度，可以計算壽命。

- 溫度列設置為 0，依據 FE 分析設定的溫度。



載荷的形式



固定載荷

評估 一

- 設定載荷類型 Hybrid 。

Edit Load Map (LoadProviderHybrid)

Edit Loading | Advanced

Loading Type: Hybrid 混合載荷類型

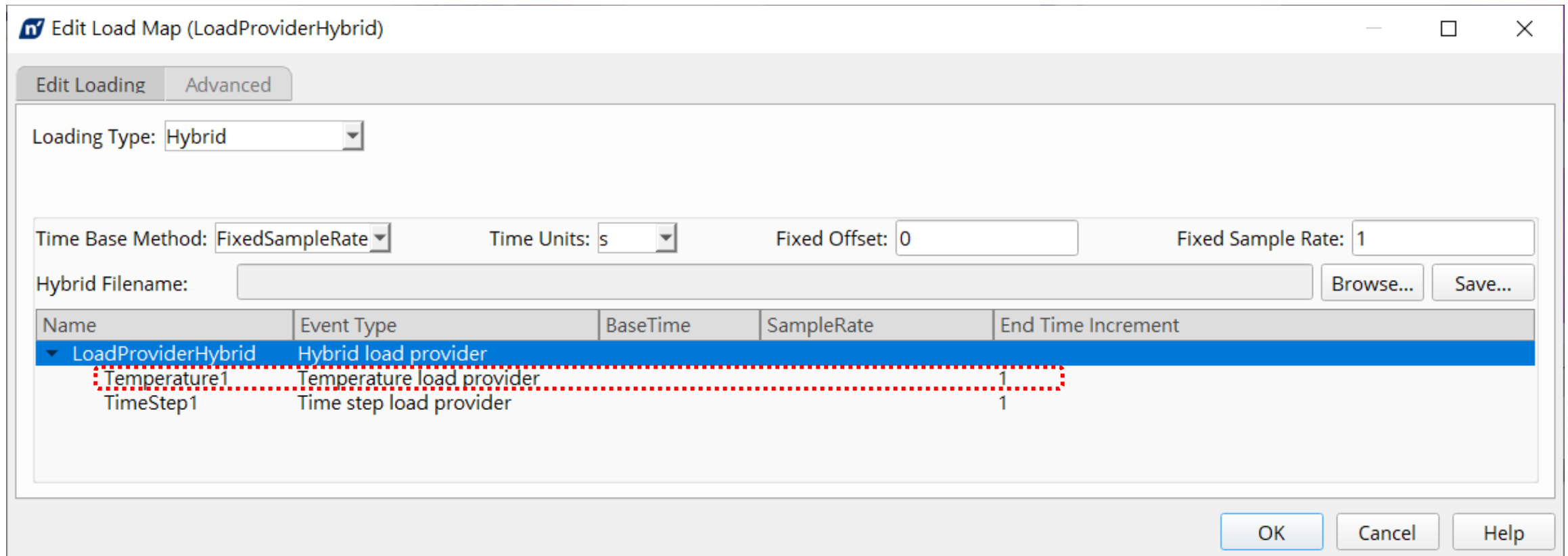
Time Base Method: FixedSampleRate | Time Units: s | Fixed Offset: 0 | Fixed Sample Rate: 1

Hybrid Filename: Browse... Save...

Name	Event Type	BaseTime	SampleRate	End Time Increment
LoadProviderHybrid	Hybrid load provider			

OK Cancel Help

- 設定溫度相關的步階(穩態熱傳 TIME STEP)。



載荷類型：溫度 TIME STEP

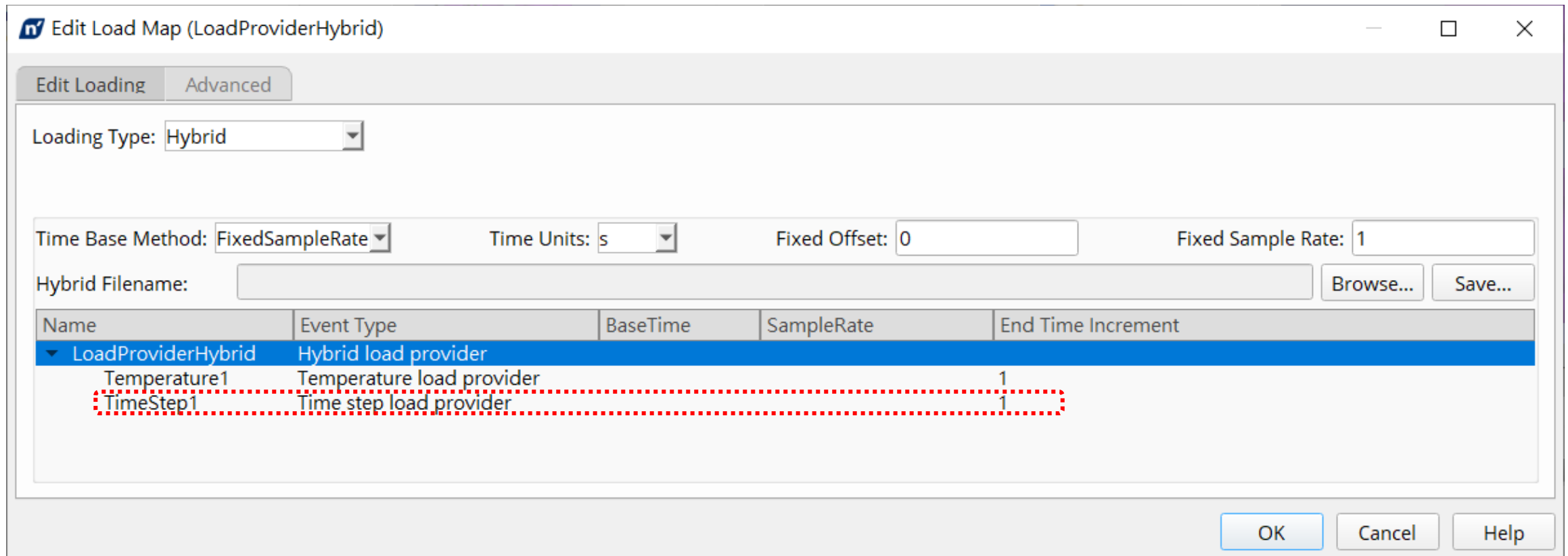
- 4 個工況，時間歷程 6,000 秒，要評估“最壞情況”，將 Mode 改為 Max。

The screenshot displays the 'Edit Load Map (Temperature1)' window in Ansys. The 'Advanced' tab is active, showing 'Loading Type: Temperature' and 'Temperature Step Descriptions: All'. The 'Mode' dropdown is set to 'Max', highlighted with a red dashed box. A red box also highlights the text '最高溫度(無溫升溫降)'. Below, a table lists four temperature steps with their descriptions and times.

Temperature Step	Description	Time
1	1 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1
2	2 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1200
3	3 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	5000
4	4 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	6000

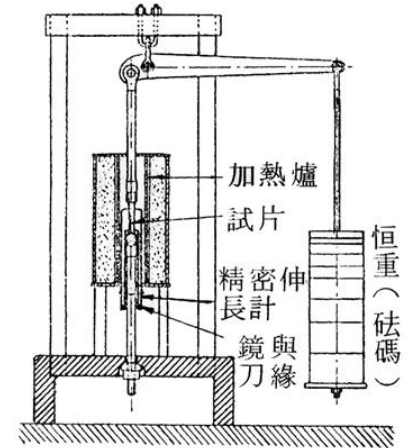
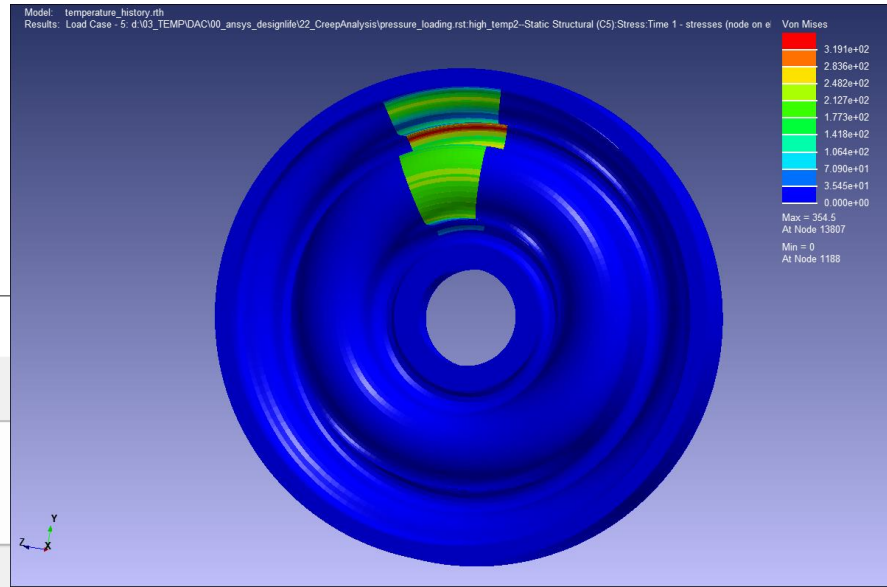
Below the table are four simulation result images of a circular component at different time steps: 1s, 1200s, 5000s, and 6000s. Each image shows a color-coded temperature distribution on the component's surface, with a legend indicating the temperature range. The maximum temperature values for each time step are: 1s (Max = 1.12E+02), 1200s (Max = 1.12E+02), 5000s (Max = 1.12E+02), and 6000s (Max = 1.12E+02).

- 設定壓力載荷。



載荷類型：應力

- 壓力無波動 (無波動應力)



Edit Load Map (TimeStep1)

Edit Loading Advanced

Loading Type: Time Step

Configuration Method: Interactive Load Case Descriptions All

Time Step Sequence Filename:

Available FE Load Cases

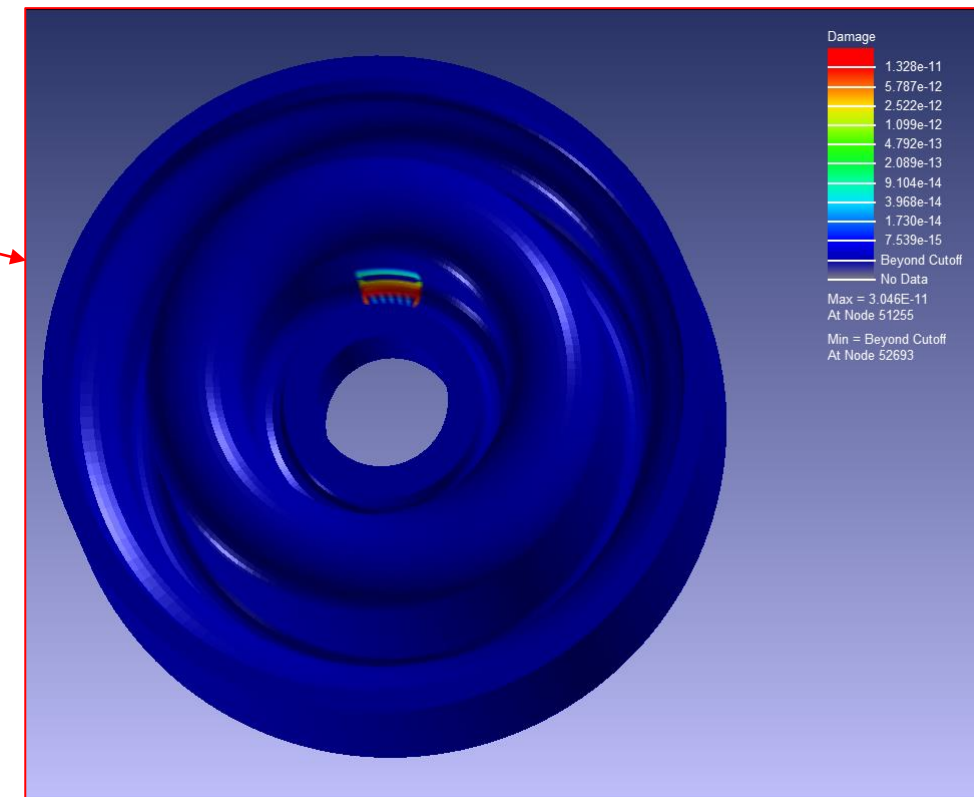
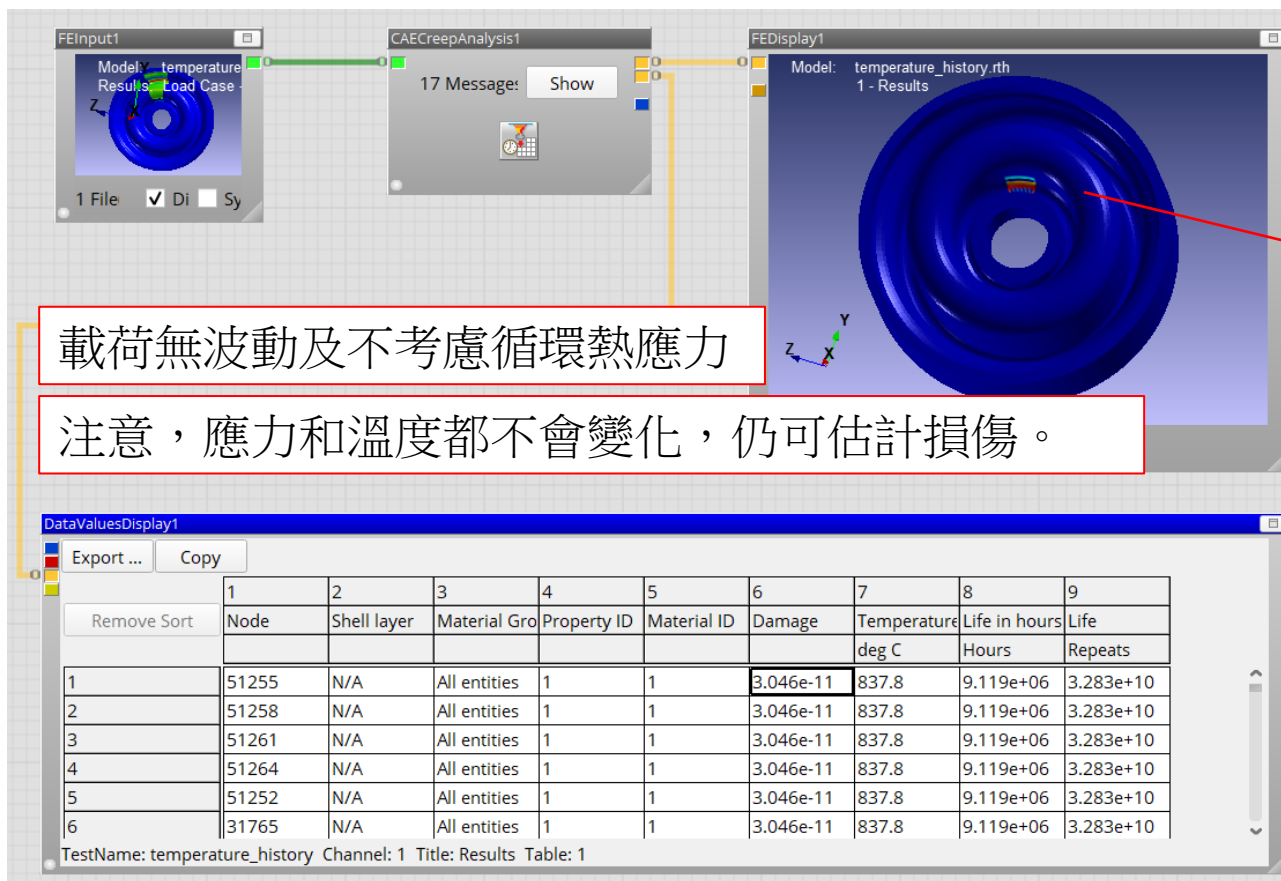
Description	>	>>	<	<<
1 - high_temp3--Static Structural 2 (E5):Stress:Time 1				
2 - high_temp3--Static Structural 2 (E5):Stress:Time 1200				
3 - high_temp3--Static Structural 2 (E5):Stress:Time 5000				
4 - high_temp3--Static Structural 2 (E5):Stress:Time 6000				
5 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnaly...				

Load Case Assignments

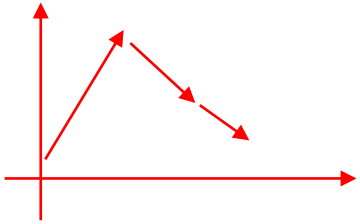
Time Step	Description	Time	Scale Factor
1	5 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnal	1	1

OK Cancel Help

- 在最壞情況下，純壓力載荷、等溫下的潛變損傷並不顯著。



載荷的形式



來自於軟體 TIME STEP 的加載

評估 二

- 設定載荷類型 Hybrid 。

Edit Load Map (LoadProviderHybrid)

Edit Loading Advanced

Loading Type: Hybrid 混合載荷類型

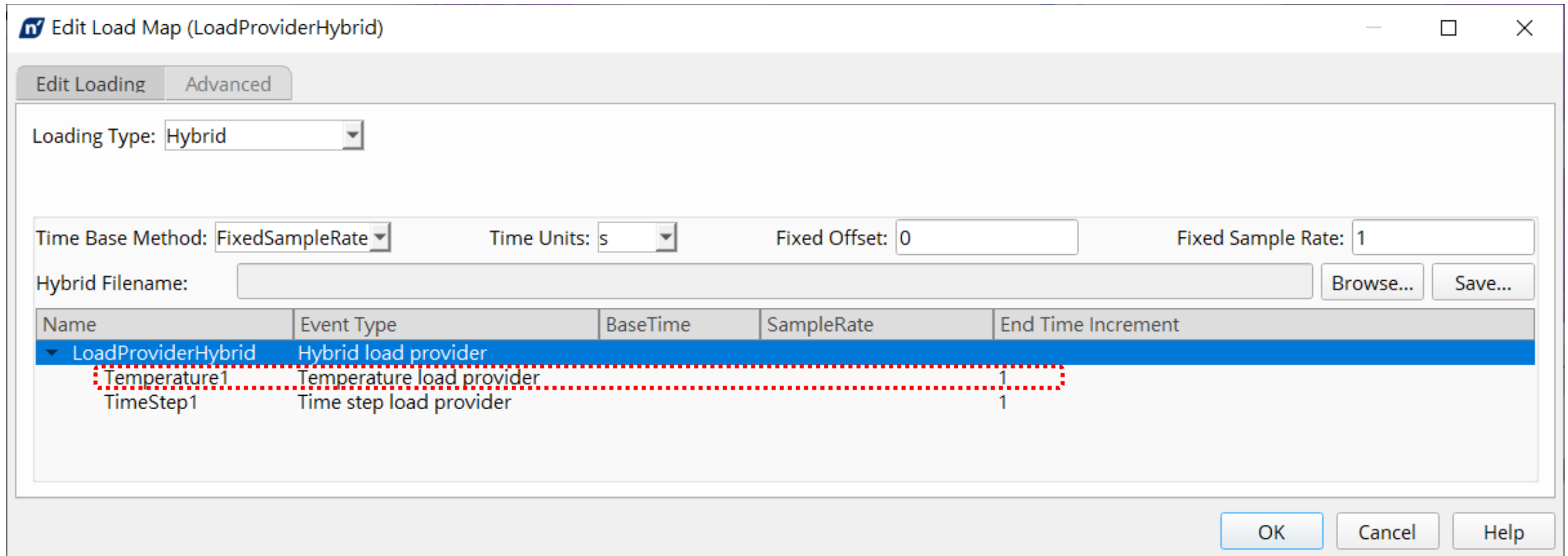
Time Base Method: FixedSampleRate Time Units: s Fixed Offset: 0 Fixed Sample Rate: 1

Hybrid Filename: Browse... Save...

Name	Event Type	BaseTime	SampleRate	End Time Increment
LoadProviderHybrid	Hybrid load provider			

OK Cancel Help

- 設定溫度相關的步階(穩態熱傳 TIME STEP)。



載荷類型：溫度 TIME STEP

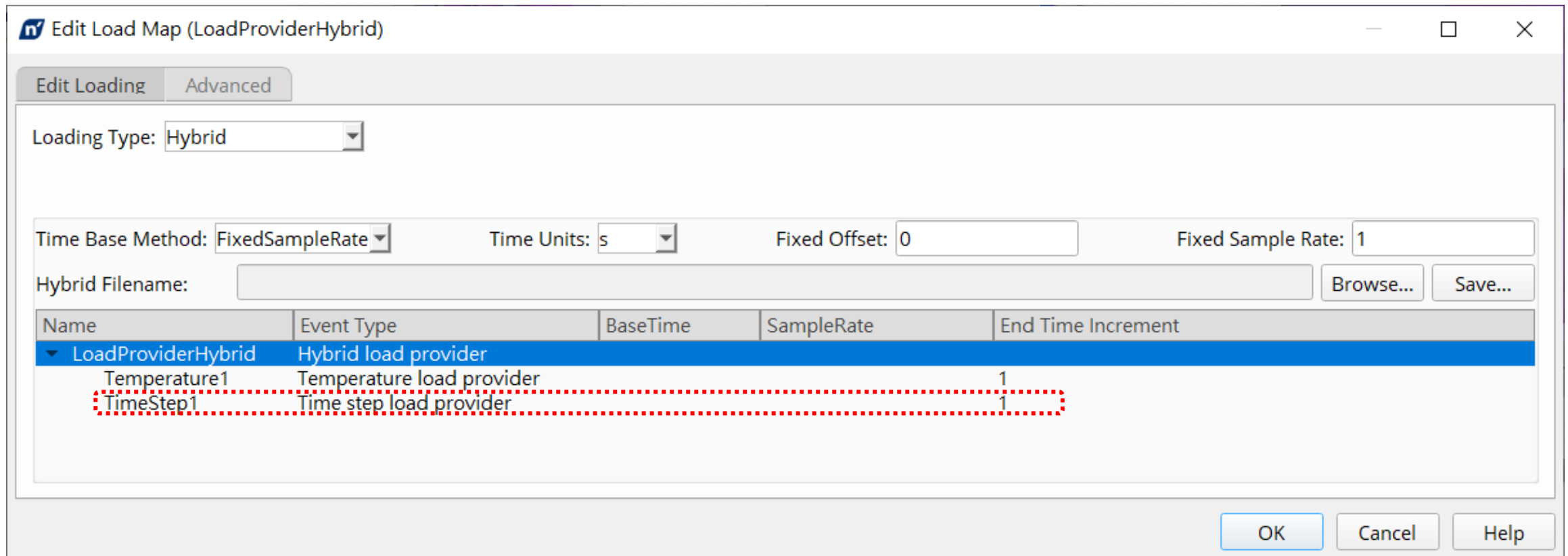
- 4 個工況，時間歷程 6,000 秒，要評估“最壞情況”，將 Mode 改為 Max。

The screenshot displays the ANSYS 'Edit Load Map (Temperature1)' window. The 'Advanced' tab is active, showing 'Loading Type: Temperature' and 'Temperature Step Descriptions: All'. A graph plots 'Temperature (C)' vs 'Time (sec)', showing 'ID Temperature' (blue) peaking at 900°C at 1200s and 'OD Temperature' (red) peaking at 100°C at 1200s. The 'Mode' dropdown is set to 'Max'. Below the graph is a table of 'Temperature Step Assignments':

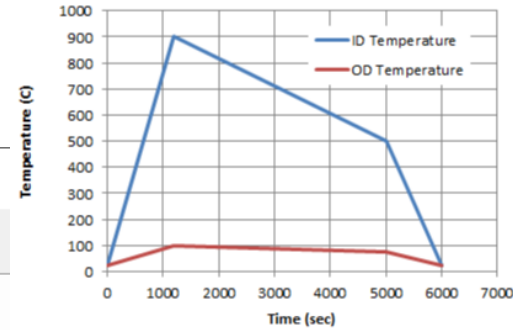
Temperature Step	Description	Time
1	1 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1
2	2 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1200
3	3 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	5000
4	4 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	6000

At the bottom, four circular cross-sections of a component are shown at different time steps: 1s, 1200s, 5000s, and 6000s. Each image includes a temperature scale legend. The 1s image shows a maximum temperature of 1.700e+01. The 1200s image shows a maximum of 9.100e+02. The 5000s image shows a maximum of 1.000e+03. The 6000s image shows a maximum of 6.120e+01.

- 設定熱應力相關的載荷歷程。



- 取消壓力載荷，僅考慮施加循環熱應力對潛變的影響



Edit Load Map (TimeStep1)

Edit Loading | Advanced

Loading Type: Time Step

Configuration Method: Interactive | Load Case Descriptions: All

Time Step Sequence Filename: [Browse...]

Available FE Load Cases

Description
1 - high_temp3--Static Structural 2 (E5):Stress:Time 1
2 - high_temp3--Static Structural 2 (E5):Stress:Time 1200
3 - high_temp3--Static Structural 2 (E5):Stress:Time 5000
4 - high_temp3--Static Structural 2 (E5):Stress:Time 6000
5 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\pressure_loadin...

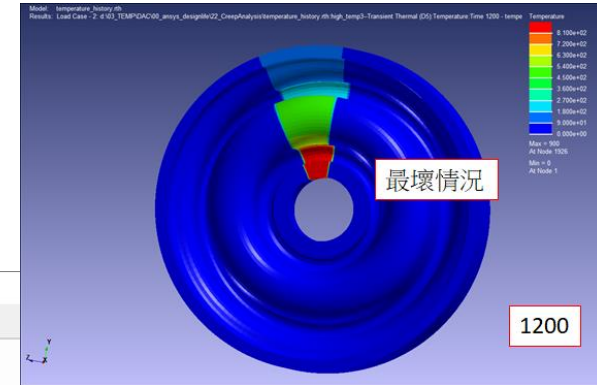
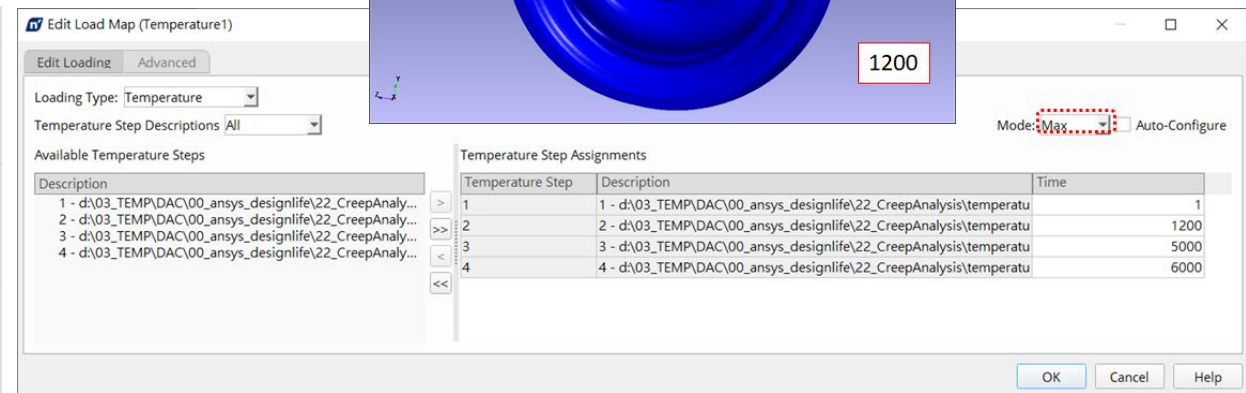
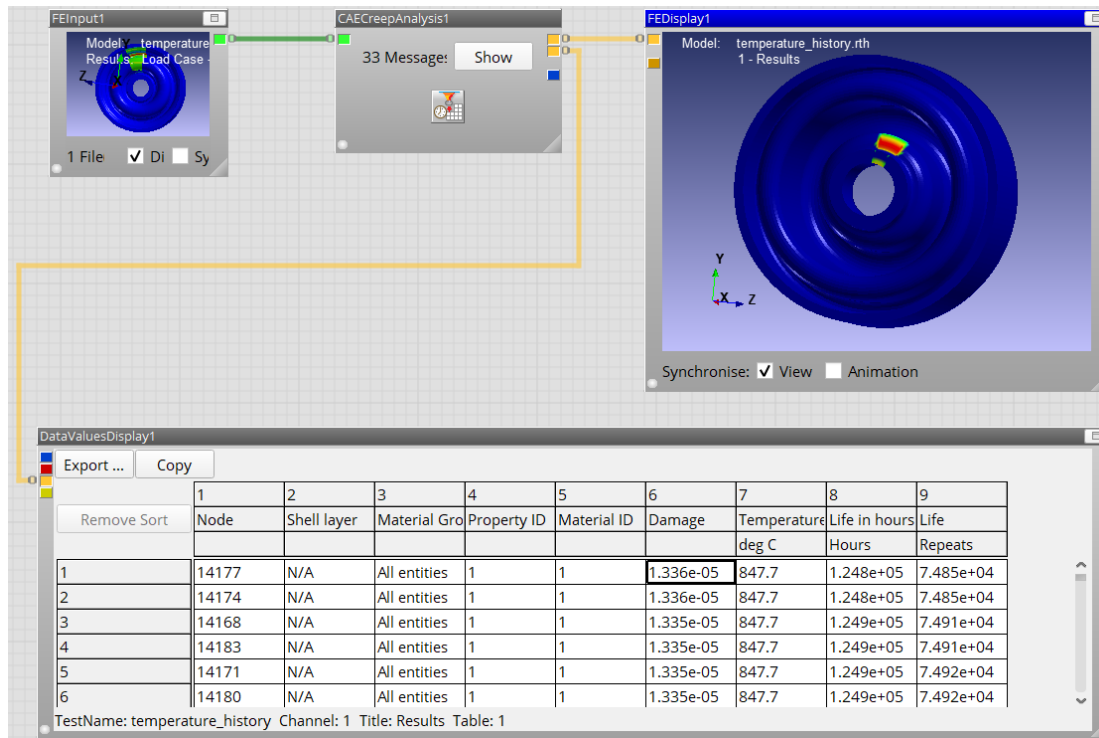
Load Case Assignments

Time Step	Description	Time	Scale Factor
1	1 - high_temp3--	1	1
2	2 - high_temp3--	1200	1
3	3 - high_temp3--	5000	1
4	4 - high_temp3--	6000	1

熱應力

OK | Cancel | Help

- 在最高溫度下，循環熱應力的潛變損傷。



At Node 14174 and is 1.248E5 hours.
This is close to 14 years

評估三

- 設定載荷類型 Hybrid 。

Edit Load Map (LoadProviderHybrid)

Edit Loading | Advanced

Loading Type: Hybrid 混合載荷類型

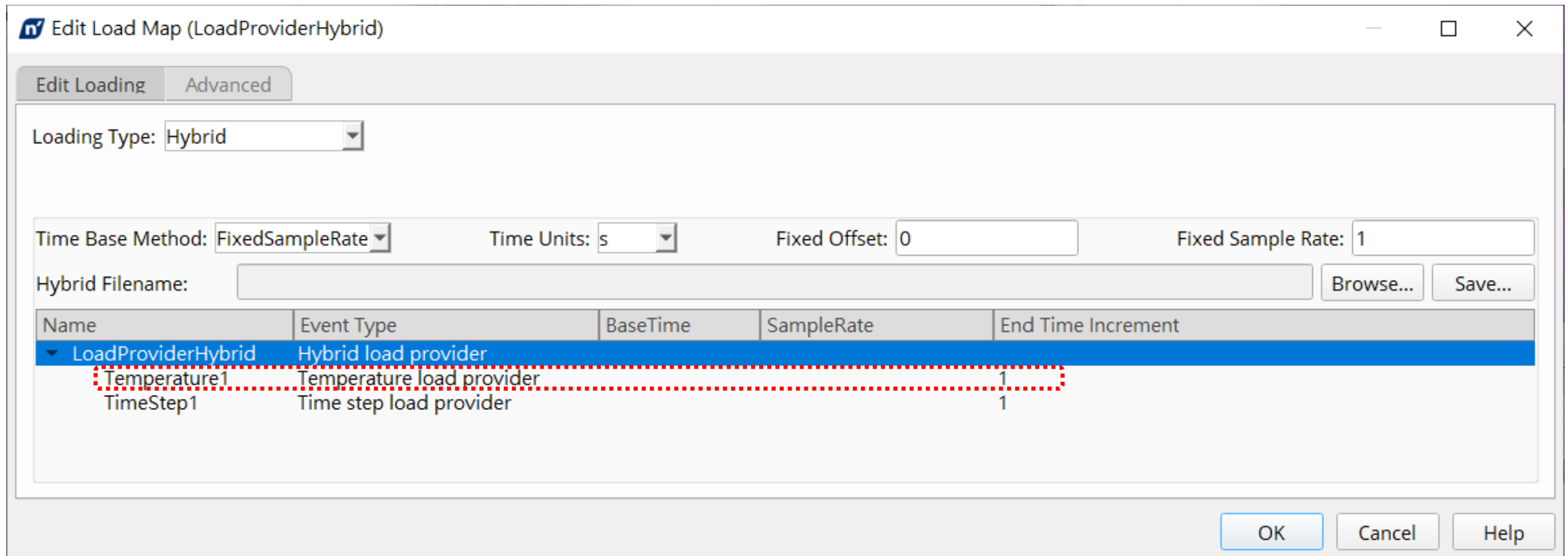
Time Base Method: FixedSampleRate | Time Units: s | Fixed Offset: 0 | Fixed Sample Rate: 1

Hybrid Filename: Browse... Save...

Name	Event Type	BaseTime	SampleRate	End Time Increment
▼ LoadProviderHybrid	Hybrid load provider			

OK Cancel Help

- 設定溫度相關的步階(穩態熱傳 TIME STEP)。



載荷類型：溫度

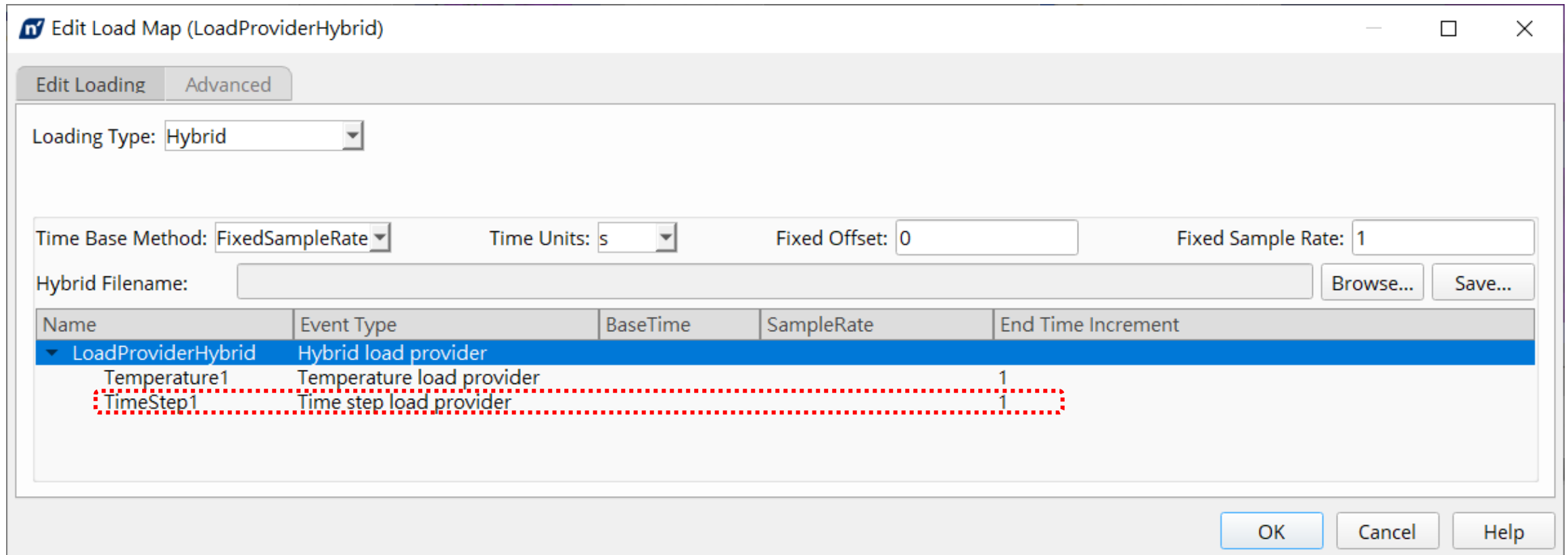
- 4 個工況，時間歷程 6,000 秒，評估“溫度歷程”，將 Mode 改為 All。

The screenshot displays the ANSYS software interface for configuring a temperature load map. The 'Edit Load Map (Temperature1)' window is open, showing the 'Advanced' tab. The 'Loading Type' is set to 'Temperature' and 'Temperature Step Descriptions' is set to 'All'. Below this, a table lists 'Available Temperature Steps' with descriptions and time points. To the right, a 'Temperature Step Assignments' table shows the selected steps and their corresponding times. A line graph plots 'Temperature (C)' against 'Time (sec)', showing two curves: 'ID Temperature' (blue) and 'OD Temperature' (red). The ID temperature rises to 900°C at 1000s and then gradually decreases to 0°C at 6000s. The OD temperature remains near 0°C throughout. A red box highlights the 'Mode: All' dropdown menu, with the text '考慮溫升溫降' (Consider temperature rise and fall) above it. At the bottom, four 3D models of a circular component are shown at different time points: 1s, 1200s, 5000s, and 6000s. Each model is color-coded to represent the temperature distribution, with a color scale legend on the right of each model. The temperature is highest at the top of the component and lowest at the bottom.

Temperature Step	Description	Time
1	1 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	1
2	2 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	1200
3	3 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	5000
4	4 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	6000

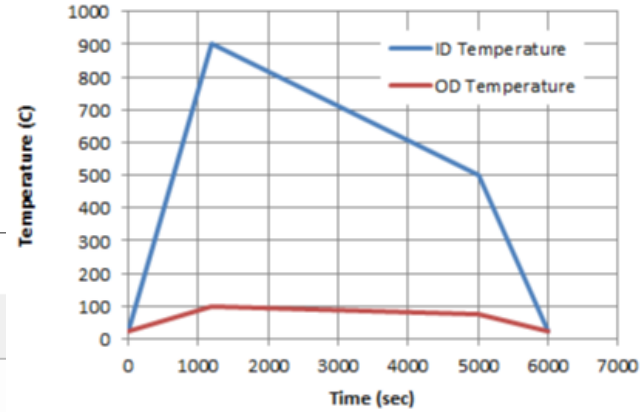
Time	Temperature (C)
0	0
1000	900
2000	800
3000	700
4000	600
5000	500
6000	0

- 設定熱應力相關的載荷歷程。



循環應力加載

- 僅考慮施加循環熱應力對潛變的影響



Edit Load Map (TimeStep1)

Edit Loading | Advanced

Loading Type: Time Step

Configuration Method: Interactive | Load Case Descriptions: All

Time Step Sequence Filename: [Browse...]

Available FE Load Cases

Description
1 - high_temp3--Static Structural 2 (E5):Stress:Time 1
2 - high_temp3--Static Structural 2 (E5):Stress:Time 1200
3 - high_temp3--Static Structural 2 (E5):Stress:Time 5000
4 - high_temp3--Static Structural 2 (E5):Stress:Time 6000
5 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\pressure_loadin...

Load Case Assignments

Time Step	Description	Time	Scale Factor
1	1 - high_temp3--	1	1
2	2 - high_temp3--	1200	1
3	3 - high_temp3--	5000	1
4	4 - high_temp3--	6000	1

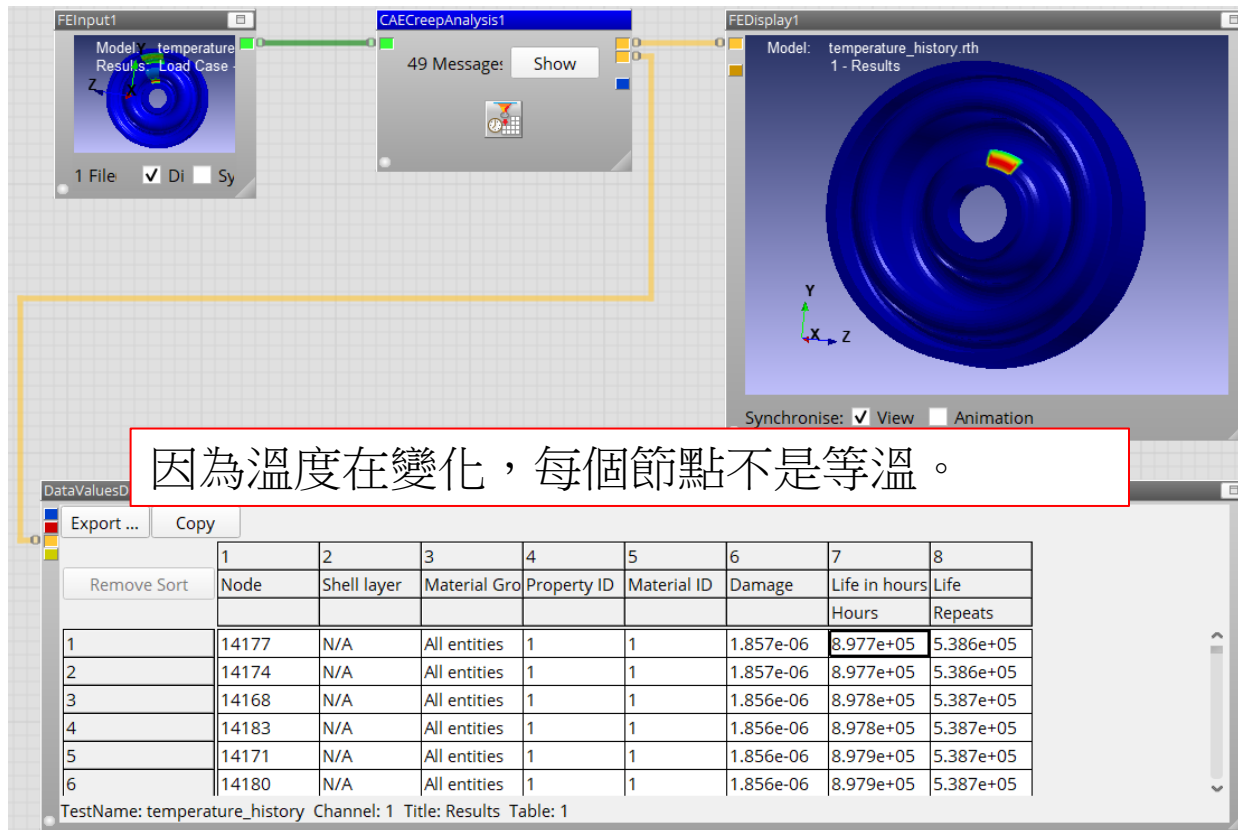
熱應力

The bottom section of the interface displays four sequential stress distribution plots of a blue flange component. Each plot shows the stress distribution at a specific time step: 1, 1200, 5000, and 6000 seconds. The stress is visualized using a color scale from blue (low stress) to red (high stress). The stress concentration is highest at the inner edge of the flange. A red arrow points to the '熱應力' (Thermal Stress) label above the table.

Auto-Configure [] | [Browse...] | [Save...]

[OK] [Cancel] [Help]

- 最壞情況損傷發生在節點 14174，為 $8.977E5$ 小時，從 14 年（使用最高溫度）增加到大約 100 年。

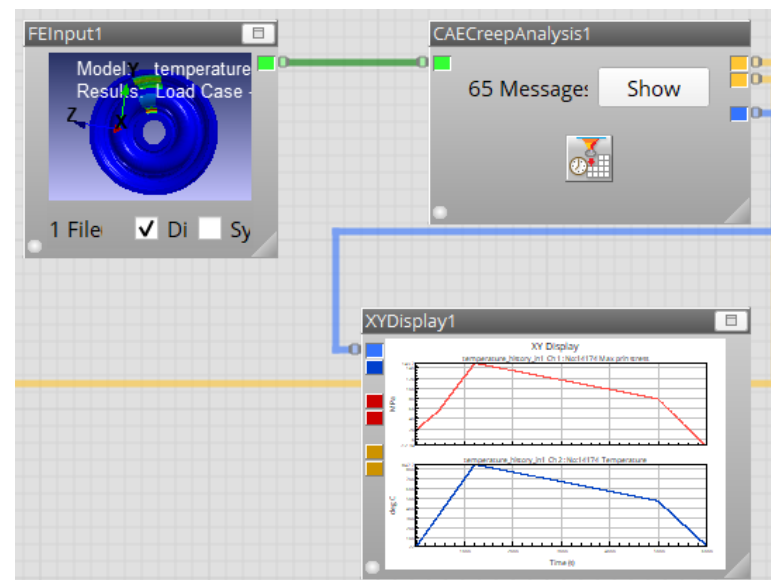


- 最大損傷發生在節點 14174，輸出結果圖。

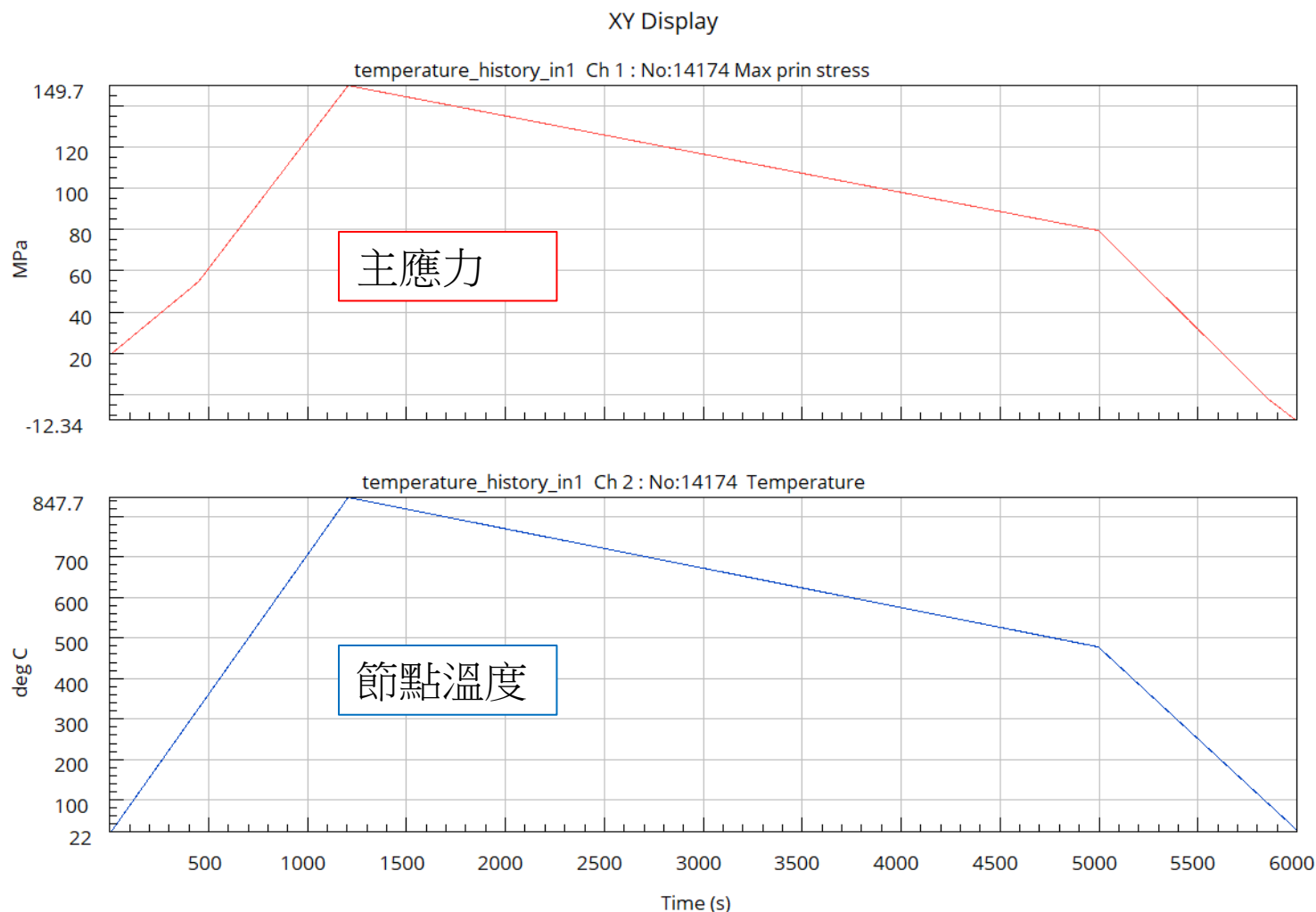
Object name	Object type
Job	Job
FE Results	
FEResultsImport	Imported FE results
ResultsSet	FE import results set
AnalysisGroup	FE import analysis group
Loading	
LoadProviderHybrid	Hybrid load provider
HybridItem1	Hybrid item
HybridItem2	Hybrid item
Materials	
MaterialMap	Material map
MaterialDatenbaseCode1	nCode material database
Analysis Runs	
Run1	Run
AnaDef	Analysis definition
CreepEngine	Creep analysis engine
TimeSeriesOutputPipe1	Time series output to pipe
MaterialMap	Material map
DefaultMaterial	Material map entry
EntitySubsetStatic1	Static entity subset
Results Handlers	
ResultsHandler	FER results handler
Post Processors	
Compressed results (for display)	Pipe post processor
Full results (tabulate)1	Pipe post processor
Entity Subsets	
EntitySubsetStatic1	Static entity subset
Unattached Items	
TemperatureLoadProvider	Temperature load provider
LoadProviderTimeStep1	Time step load provider

Object Name: EntitySubsetStatic1 (Static entity subset)

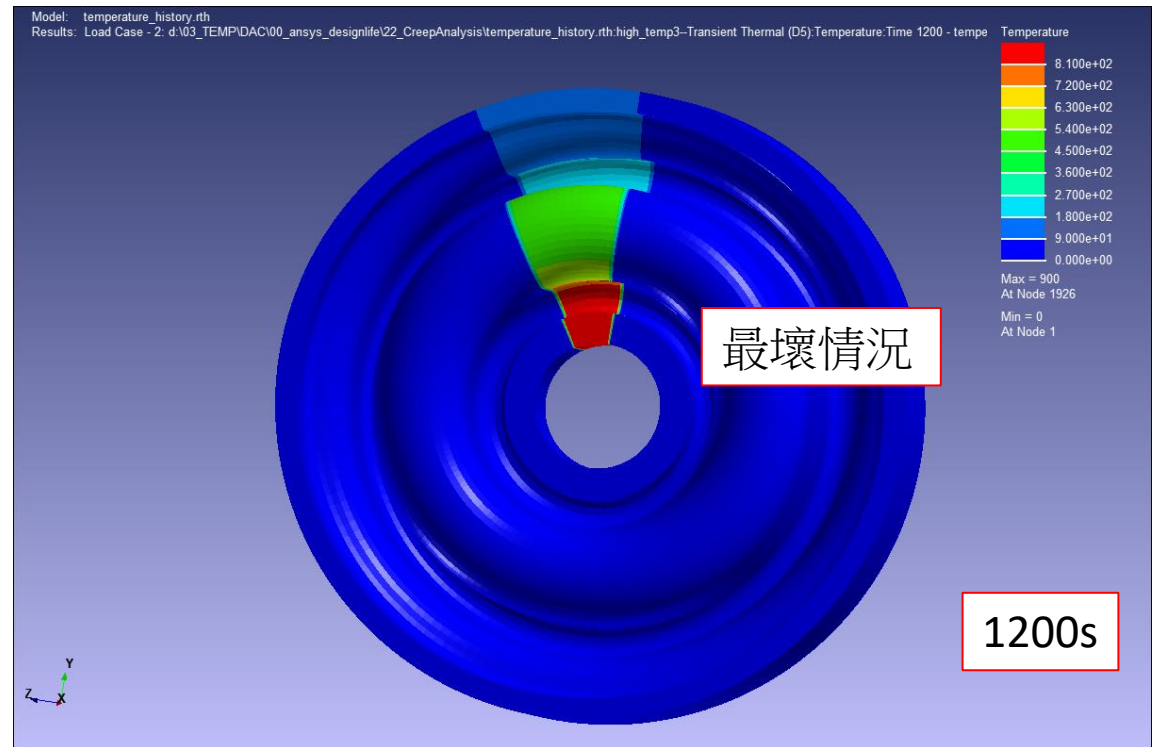
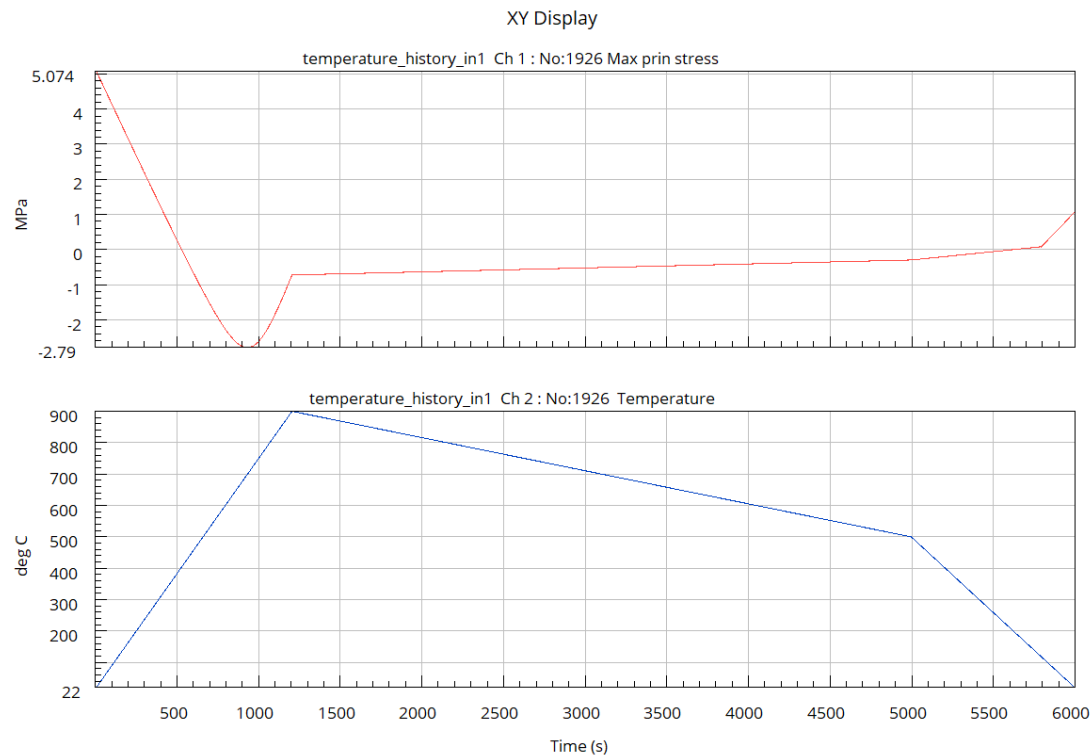
Name	Value	Description
General		
EntitySource	FromIDs	The source for the list of entity IDs.
ProcessInOrder	False	Set True to cause subset entities to be processed in order
Active	True	Set False to cause this subset to not be processed.
Ids		
Ids	14174	The list of entity ids
EntityType	Nodes	The type of entity that this subset lists



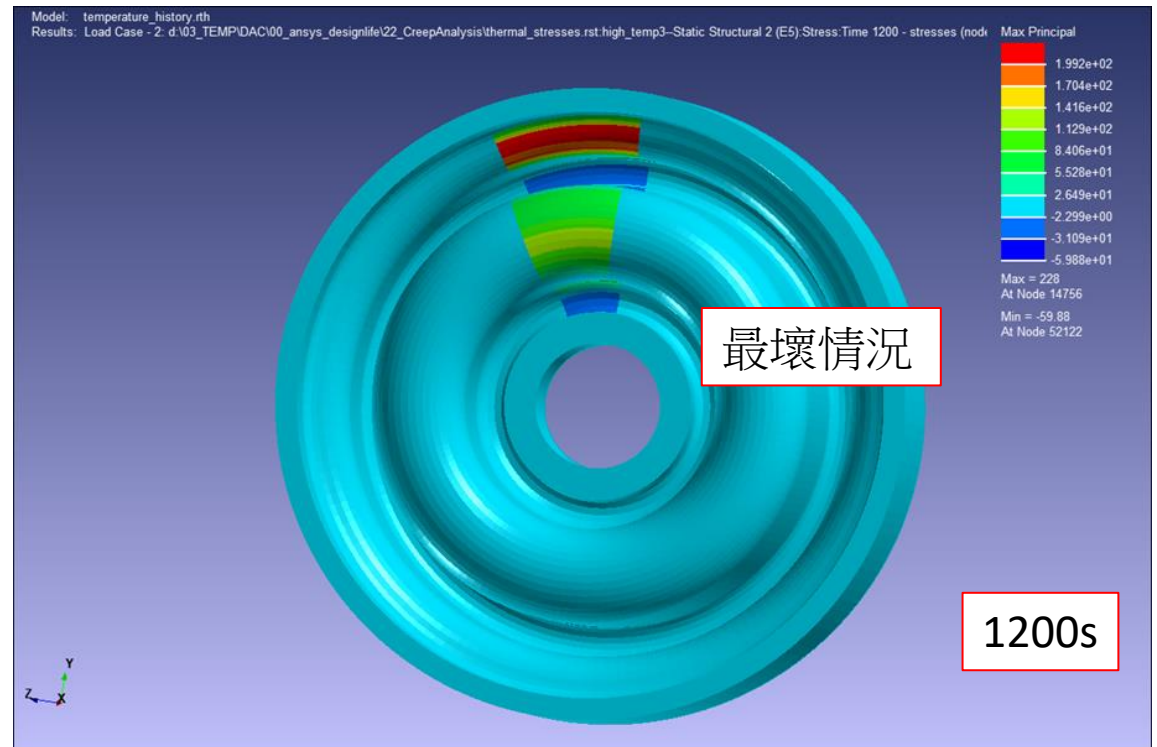
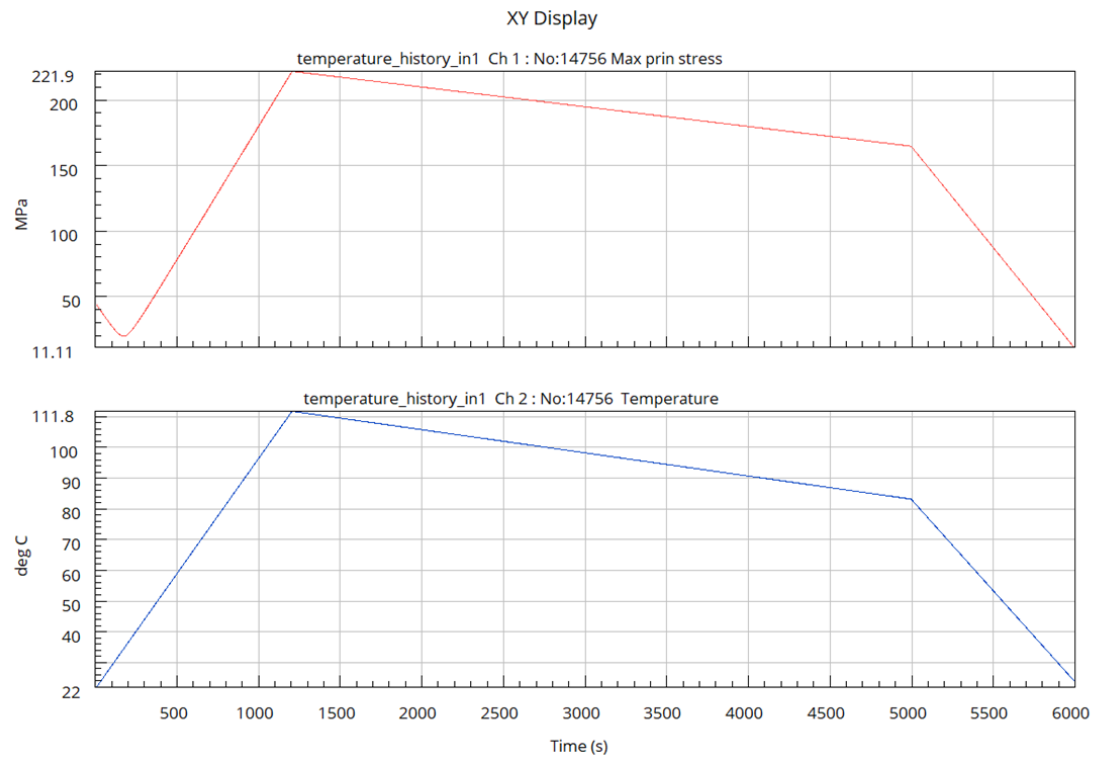
- 節點(14174)溫度及主應力都相對較高。
- 熱應力和溫度是同相的，雖然峰值應力和峰值溫度同時出現，但在很長一段時間內，應力和溫度都會低於最大值，因此潛變損傷會低於最大值(皆使用最高溫度時)。



- 節點溫度最高，主應力很小。



- 節點主應力最大，溫度111.8 度。



內容大綱：

1. 前言 nCode Highlight
2. 溫度相依的疲勞 Temperature dependent fatigue
3. 潛變斷裂 Creep rupture
4. 高溫結構疲勞分析 Thermo-mechanical fatigue

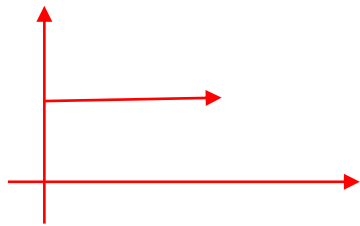
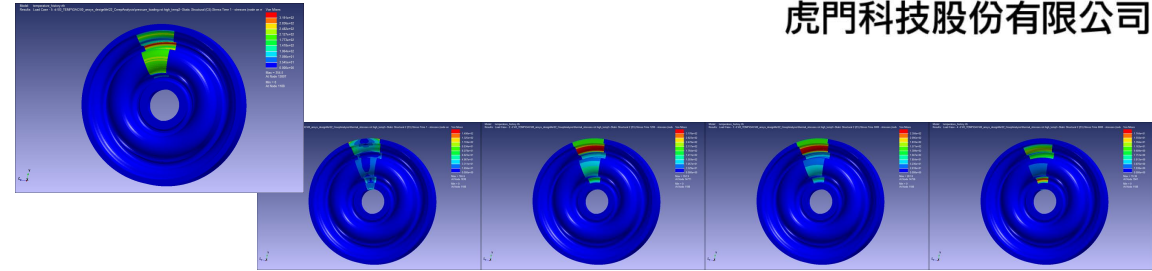
- 零件的疲勞壽命會受到其溫度環境的影響。
- 溫度變化不僅會引起應力循環，從而導致疲勞損壞，而且材料的疲勞強度也可能隨溫度而變化。
- 考慮到這一點，我們需要能夠為模型上的每個位置，分配適合其溫度歷程的疲勞屬性。
- 還需要一種有效的方法，將熱應力歷程與其它來源的應力結合起來，這些來源可能在不同的時間尺度上。

- 混合載荷程序提供兩種功能。它允許將溫度和應力匹配在一起，以進行與溫度相關的疲勞計算。另提供了一種透過組合不同類型的負載程序，來創建壓力時間歷程的替代方法。
- 一個混合載荷，必須至少有一個子載荷，可以是以下類型：
 - 1) 時間步長
 - 2) 等幅
 - 3) 時間序列

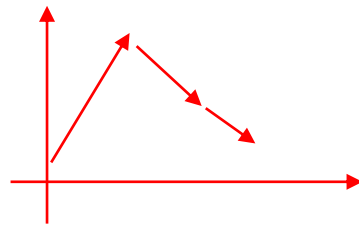
- 1) Time Step
- 2) Constant Amplitude
- 3) Time Series

分析前的考量

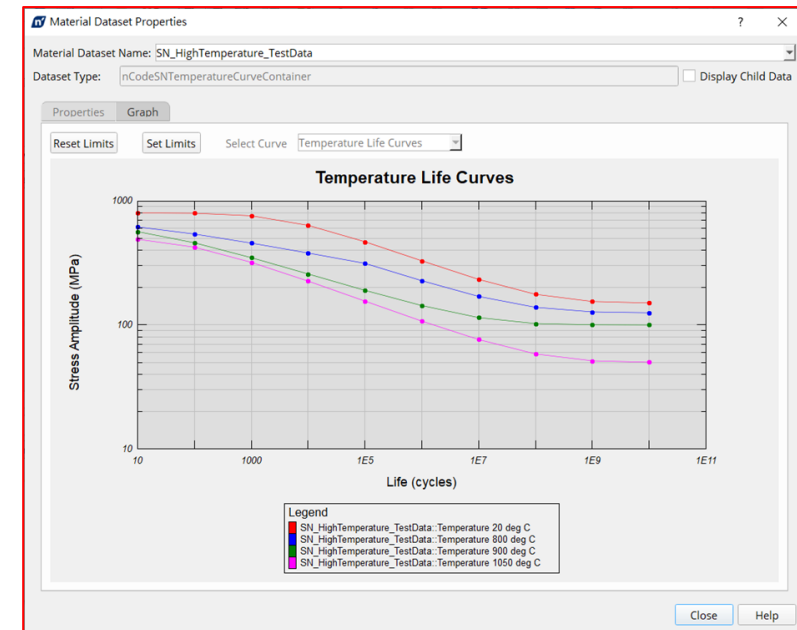
1. 應力結果_ANSYS 應力分析，包含熱應力
2. SN 曲線圖 → 高溫SN曲線圖
3. 載荷的形式



固定載荷

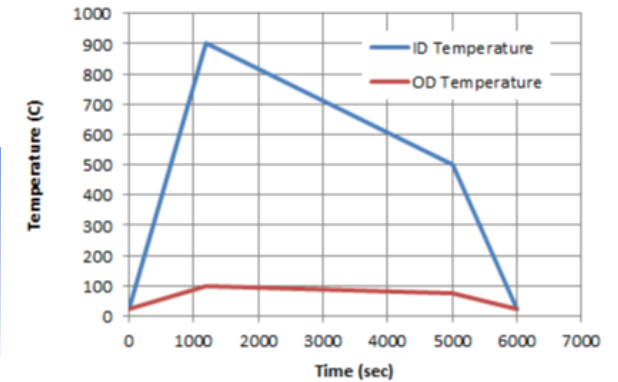
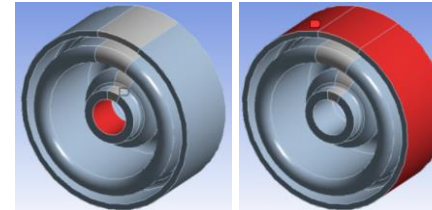
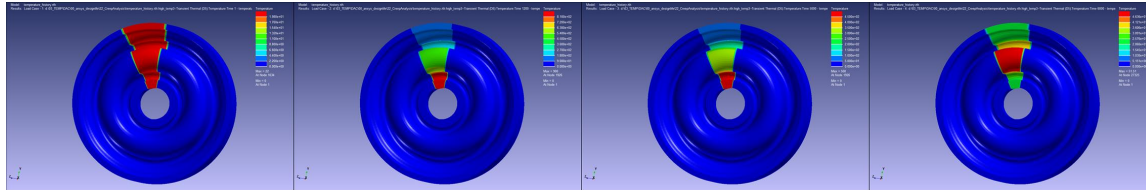


來自於軟體 TIME STEP 的加載

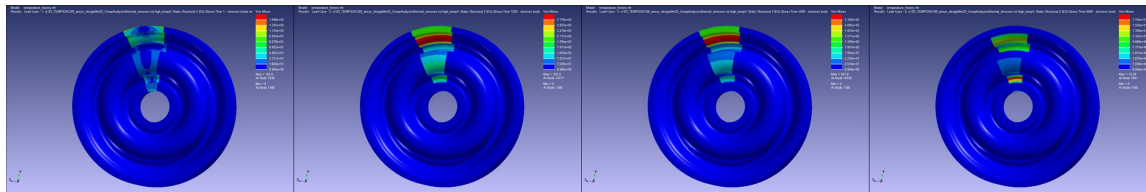


➤ 有限元分析分三個階段進行，產生三組 ANSYS 結果檔。

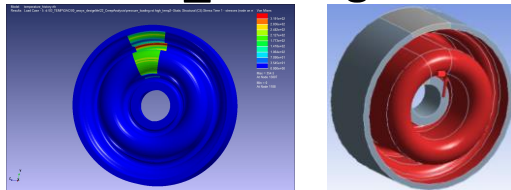
- Temperature_history.rth 包含時間 1、1200、5000 和 6000 秒的模型溫度。



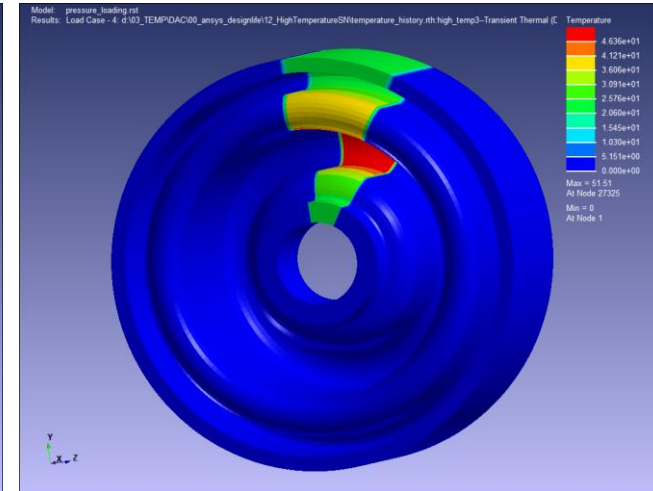
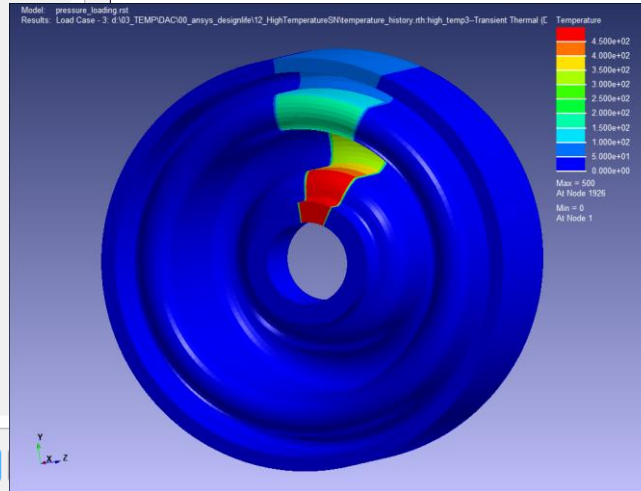
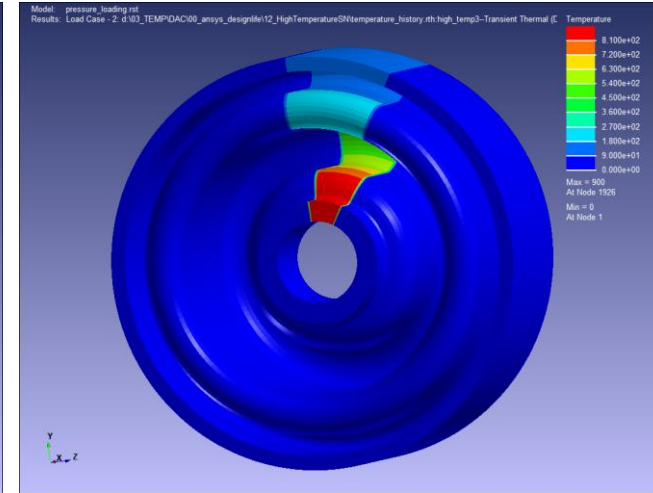
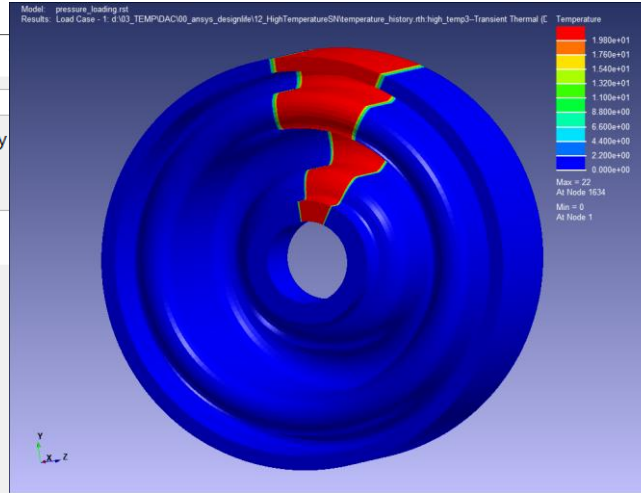
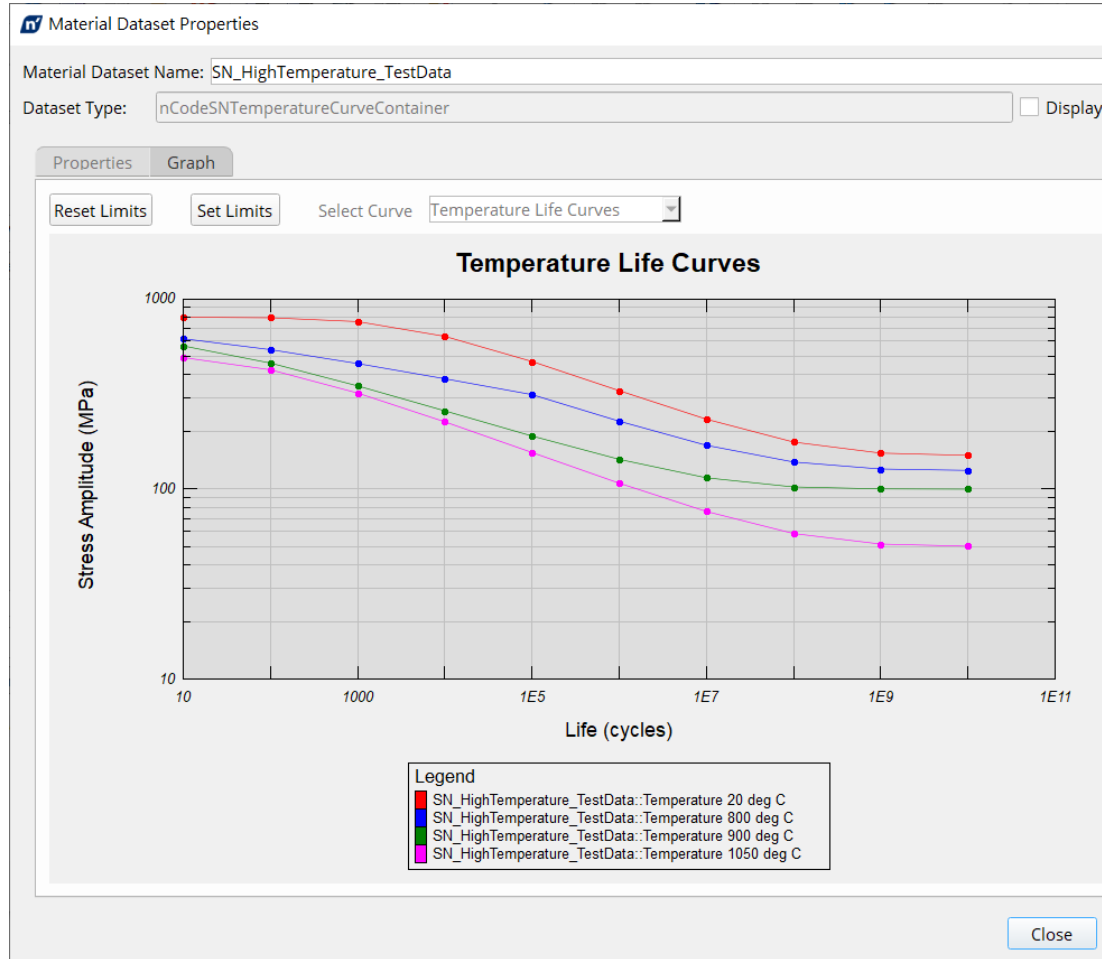
- Thermal_stresses.rst 包含四組波紋管在時間 1、1200、5000 和 6000 秒處的應力，這是由於 temperature_history.rth 中描述的溫度歷程引起的熱膨脹。



- Pressure_loading.rst 包含由施加到波紋管一個面的 5 MPa 壓力載荷引起的單一應力狀態。



高溫的疲勞曲線



- 包含溫度的結果

DesignLife Configuration Editor

Object Name: ResultsSet (FE import results set)

Name	Value	Description
General		
Description		Results set description
FEFileNames	d:\03_TEMP\DAC\01	Specify FE files to import results from
VibrationOffset	False	Static offset for vibration analysis
IncludeTemperatures	True	Whether to include temperature results
IncludeEquivalentPlasticStrain	False	Whether to include equivalent plastic strain results (Dang Van only)
IncludeDisplacements	None	Specifies whether to write nodal displacements to the intermediate file
FEResults	Standard	Specifies the type of results to access in the FE results file
LoadCaseSelection		
SelectionGroupType	FEInput	Specifies the method to use for load case selection

Show editor when glyph is double clicked

OK Cancel Help

設定溫度選項

DesignLife Configuration Editor

Object name	Object type
Job	Job
FE Results	
FEResultsImport	Imported FE results
ResultsSet	FE import results set
AnalysisGroup	FE import analysis group
Loading	
LoadProviderHybrid	Hybrid load provider
Materials	
MaterialMap	Material map
MatDB	nCode material database
Analysis Runs	
Run1	Run
AnaDef	Analysis definition
SNEngine	Standard SN analysis engine
MaterialMap	Material map
Results Handlers	
ResultsHandler	FER results handler
Compressed results (for display)	Pipe post processor
Full results (tabulate)	Pipe post processor
Post Processors	
Compressed results (for display)	Pipe post processor
Full results (tabulate)	Pipe post processor

Object Name: SNEngine (Standard SN analysis engine)

Name	Value	Description
General		
SNMethod	MultiTemperatureCurve	The method used to calculate damage from a stress cycle
CustomSNMethod	None	Specifies the customised SN method to use
CombinationMethod	CriticalPlane	The method used to combine component stresses/strains
MeanStressCorrection	Goodman	The method used to correct the damage calculation for mean stress
InterpolationLimit	UseMaxCurve	Multicurve material interpolation limit
MultiAxialAssessment	Standard	Whether to perform assessment of the multi-axial stress state
CertaintyOfSurvival	50	Required certainty of survival (%) on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	True	Whether to output max and min stresses
SmallCycleCorrection	None	Adjusts materials data in the high cycle regime.
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZero
BackCalculation		
BackCalcMode	None	Whether to perform a back-calculation or not
TargetDamage	1E-6	Target damage for back calculation
BackCalcAccuracy	5	The accuracy of the back calculation
BackCalcMaxScale		The max scale factor to allow in back calculation
BackCalcMinScale		The min scale factor to allow in back calculation
Temperature		
TemperatureSelection	AllAvailable	How to process temperature data
TemperatureInterpolationLimit	UseMaxCurve	Whether to extrapolate above the highest temperature curve

Show editor when glyph is double clicked

OK Cancel Help

- 設定載荷類型 Hybrid 。

Edit Load Map (LoadProviderHybrid)

Edit Loading | Advanced

Loading Type: Hybrid 混合載荷類型

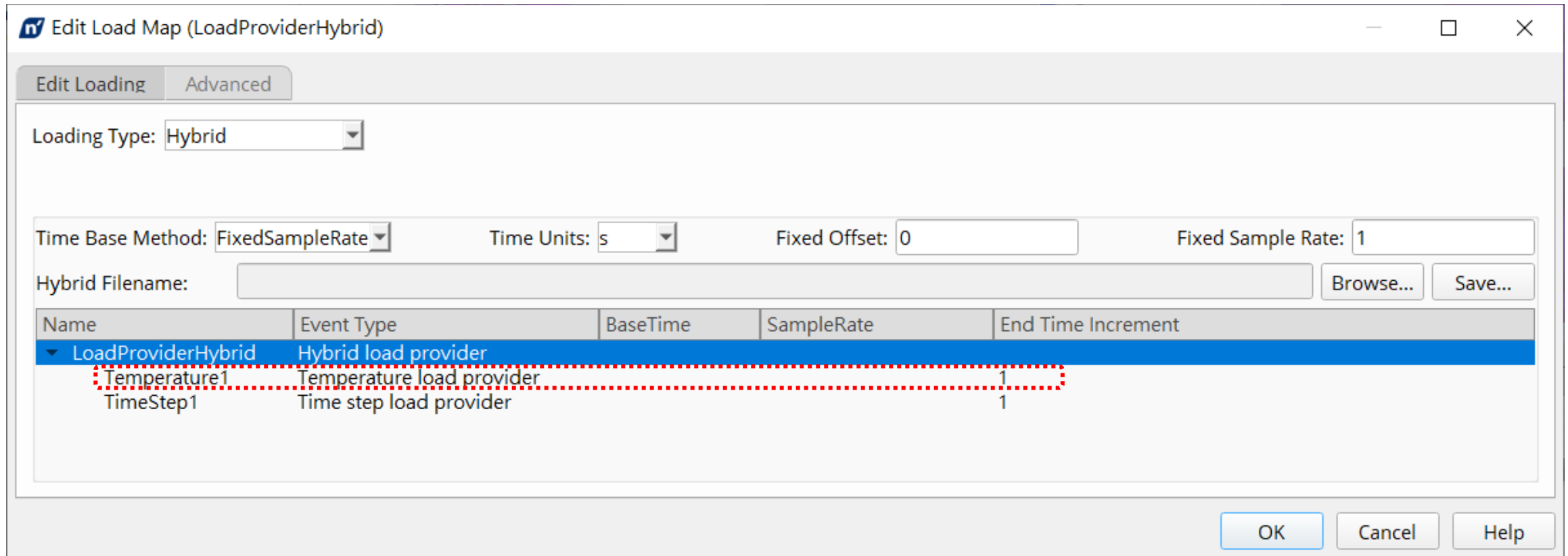
Time Base Method: FixedSampleRate | Time Units: s | Fixed Offset: 0 | Fixed Sample Rate: 1

Hybrid Filename: Browse... Save...

Name	Event Type	BaseTime	SampleRate	End Time Increment
▼ LoadProviderHybrid	Hybrid load provider			

OK Cancel Help

- 設定溫度相關的步階(穩態熱傳 TIME STEP)。



載荷類型：溫度 TIME STEP

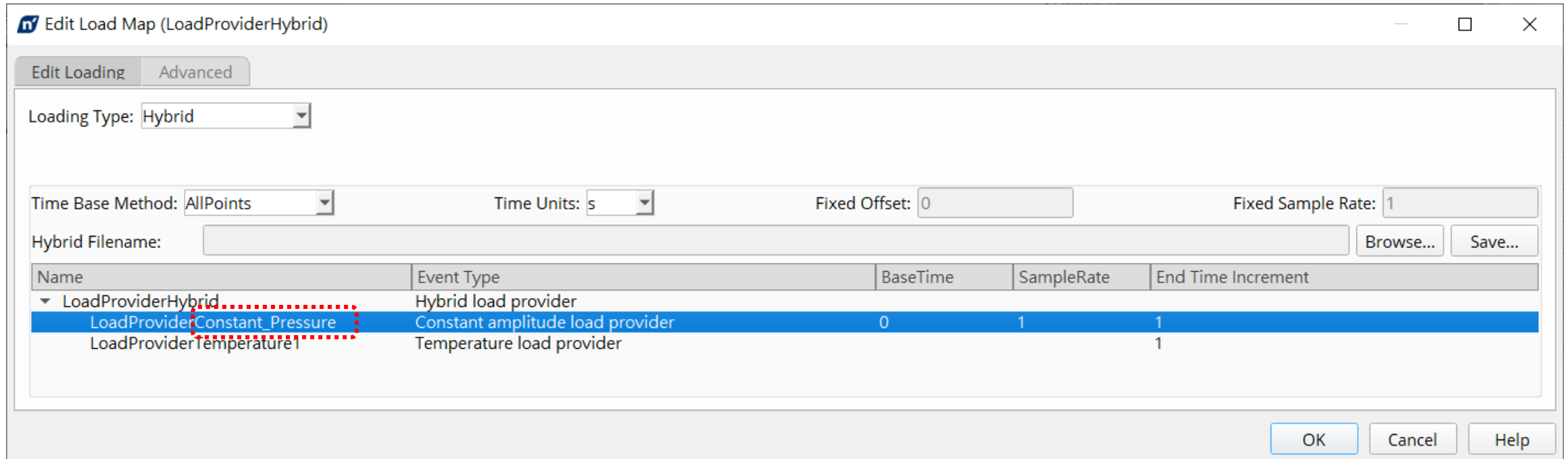
- 4 個工況，時間歷程 6,000 秒，要評估“最壞情況”，將 Mode 改為 Max。

The screenshot displays the ANSYS 'Edit Load Map (Temperature1)' window. The 'Advanced' tab is active, showing 'Loading Type: Temperature' and 'Temperature Step Descriptions: All'. A graph plots 'Temperature (C)' vs 'Time (sec)', showing 'ID Temperature' (blue) peaking at 900°C at 1200s and 'OD Temperature' (red) peaking at 100°C at 1200s. The 'Mode' dropdown is set to 'Max'. Below the graph is a table of 'Temperature Step Assignments':

Temperature Step	Description	Time
1	1 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1
2	2 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	1200
3	3 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	5000
4	4 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperatu	6000

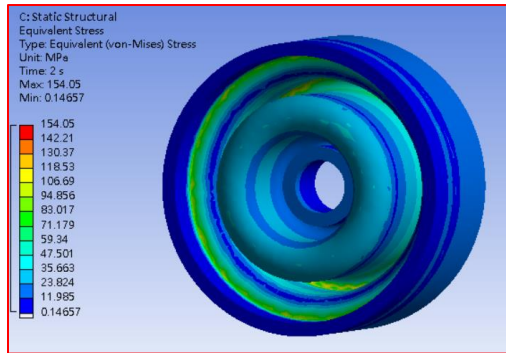
Four 3D models of a circular component are shown at different time steps: 1s, 1200s, 5000s, and 6000s. Each model has a color-coded temperature distribution. A red box highlights the '1s' model, and another red box highlights the '6000s' model. The '1s' model shows a high temperature peak (red) on the inner edge, while the '6000s' model shows a lower temperature peak (green/yellow) on the inner edge.

- 溫度和應力匹配



載荷工況(1)

- *.rst file 受壓力時的應力結果檔



Edit Load Map (LoadProviderHybrid)

Edit Loading | Advanced

Loading Type: Hybrid

Time Base Method: AllPoints | Time Units: s | Fixed Offset: 0

Hybrid Filename:

Name	Event Type	BaseTime
LoadProviderHybrid		
Temperature1		
pressure		
thermal stress		

Context Menu: Add (selected), Edit, Rename, Delete, Property Editor

Add Menu: Time series load provider, Constant amplitude load provider, Time step load provider, Temperature load provider

Edit Load Map (pressure)

Edit Loading | Advanced

Loading Type: Constant Amplitude

Load Case Descriptions: All

Available FE Load Cases

Description
1 - d:\03_TEMP\ISOThermal_nCode_files\d...
2 - d:\03_TEMP\ISOThermal_nCode_files\d...

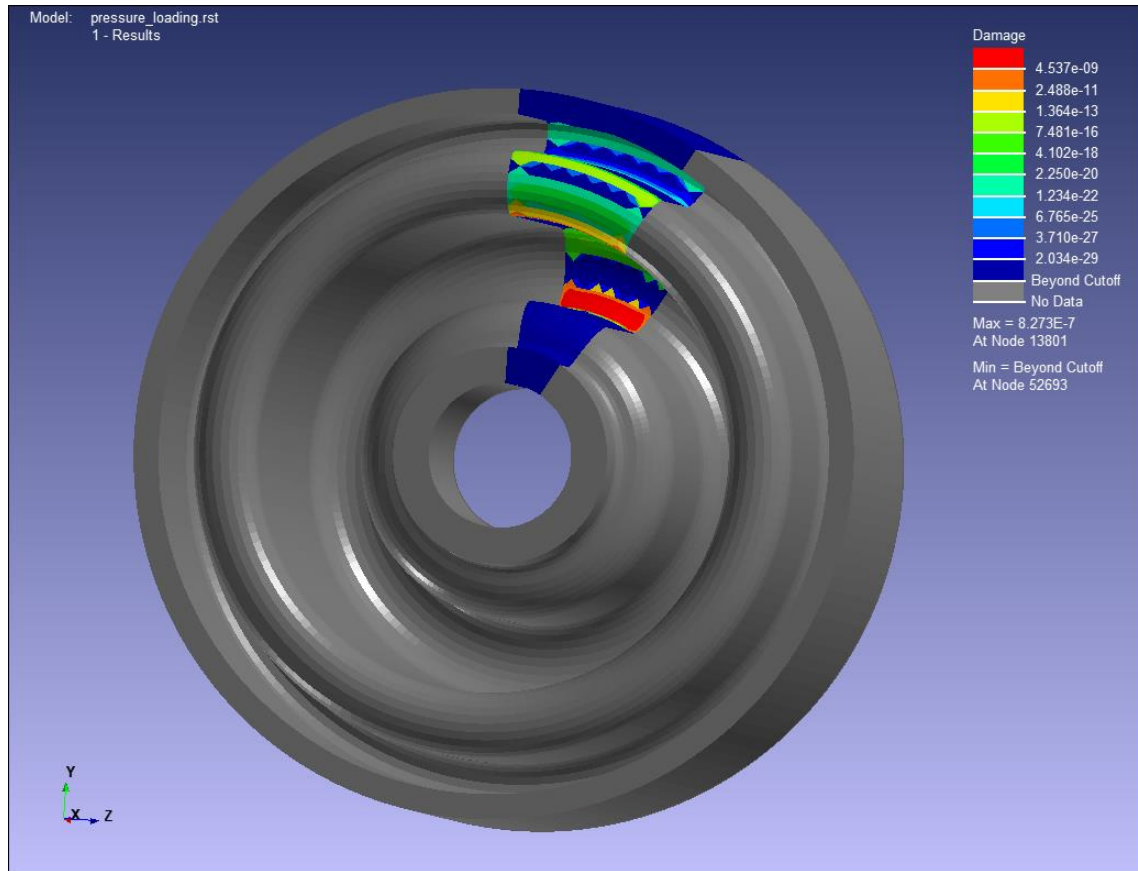
Load Case Assignments

Load Case	Description	Max Factor	Min Factor
1	2 - d:\03_TEMP\ISOThermal_nCode_files\d...	1	0

Graph: Sinusoidal wave between 2.00 and -2.00

Buttons: OK, Cancel, Help

- SN 高溫疲勞損傷



載荷類型：溫度

- 4 個工況，時間歷程 6,000 秒，評估“溫度歷程”，將 Mode 改為 All。

The screenshot displays the ANSYS software interface for configuring a temperature load map. The 'Edit Load Map (Temperature1)' window is open, showing the 'Advanced' tab. The 'Loading Type' is set to 'Temperature' and 'Temperature Step Descriptions' is set to 'All'. Below this, a table lists 'Available Temperature Steps' and 'Temperature Step Assignments'.

Temperature Step	Description	Time
1	1 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	1
2	2 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	1200
3	3 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	5000
4	4 - d:\03_TEMP\DAC\00_ansys_designlife\22_CreepAnalysis\temperature_history	6000

To the right, a graph plots 'Temperature (C)' against 'Time (sec)'. It shows two curves: 'ID Temperature' (blue) and 'OD Temperature' (red). The ID temperature rises sharply to 900°C at 1000s, then gradually decreases to 500°C at 5000s, and finally drops to 0°C at 6000s. The OD temperature remains relatively constant around 100°C throughout the duration.

Below the graph, a window titled '考慮溫升溫降' (Consider temperature rise and fall) shows the 'Mode' dropdown menu set to 'All', with 'Auto-Configure' unchecked.

At the bottom, four circular cross-sections of a component are shown, representing the temperature distribution at different time points: 1s, 1200s, 5000s, and 6000s. Each image includes a color scale legend for temperature, with red indicating the highest temperature and blue indicating the lowest. The 1s image shows a high-temperature region (red) on the top edge. As time progresses, this high-temperature region expands and then gradually cools down, as seen in the 1200s, 5000s, and 6000s images.

- 溫度和應力匹配

Temperature		
TemperatureSelection	CycleMax	How to process temperature data
TemperatureInterpolationLimit	UseMaxCurve	Whether to extrapolate above the highest temperature curve

CycleMax，這意味著每個循環使用在該應用循環期間發生的最高溫度，而不是整體最高溫度

Edit Load Map (LoadProviderHybrid)

Edit Loading Advanced

Loading Type: Hybrid

Time Base Method: AllPoints Time Units: s Fixed Offset: 0 Fixed Sample Rate: 1

Hybrid Filename: Browse... Save...

Name	Event Type	BaseTime	SampleRate	End Time Increment
LoadPr... Constant ampli... 100	Hybrid load pr...	100	0.01	100
LoadPr... Temperature lo...	Temperature lo...			1

2.00
1.50
1.00
0.50
0.00
-0.50
-1.00
-1.50
-2.00

XY Display
pressure_loading_in1 Ch 1 : No:13801 Stress at angle 0

415.5
300
200
100
0

MPa

應力波動

載荷工況(3)

- 溫度、壓應力和熱應力匹配

Edit Load Map (LoadProviderHybrid)

Edit Loading: **Advanced**

Loading Type: Hybrid

Time Base Method: AllPoints | Time Units: s | Fixed Offset: 0 | Fixed Sample Rate: 1

Hybrid Filename: [Browse... Save...]

Name	Event Type	BaseTime	SampleRate	End Time Increment
LoadProviderHybrid	Hybrid load provider			
LoadProviderTimeStep1	Time step load provider			1
LoadProviderConstant1	Constant amplitude load provider	100	0.01	100
LoadProviderTemperature1	Temperature load provider			1

XY Display

pressure_loading_in1 Ch 1 : No:13801 LoadProviderTemperature1 : Temperature (deg C)

pressure_loading_in1 Ch 2 : No:13801 LoadProviderConstant1 : X (MPa)

pressure_loading_in1 Ch 3 : No:13801 LoadProviderTimeStep1 : X (MPa)

pressure_loading_in1 Ch 1 : No:13801 Stress at angle 0 (MPa)

應力波動

- 溫度、壓應力歷程和熱應力匹配

The screenshot displays the ANSYS software interface for configuring a Hybrid load provider. The main window is titled "Edit Load Map (LoadProviderHybrid)".

Configuration Panel:

- Loading Type: Hybrid
- Time Base Method: FixedSampleRate
- Hybrid Filename: (empty)
- Fixed Sample Rate: 1

Load Case Assignments Table:

Name	Event Type	BaseTime	SampleRate	End Time Increment
LoadProviderHybrid	Hybrid load provider			
LoadProviderTimeSeries1	Time series load provider	0	0.01	100
LoadProviderTimeStep1	Time step load provider			1
LoadProviderTemperature1	Temperature load provider			1

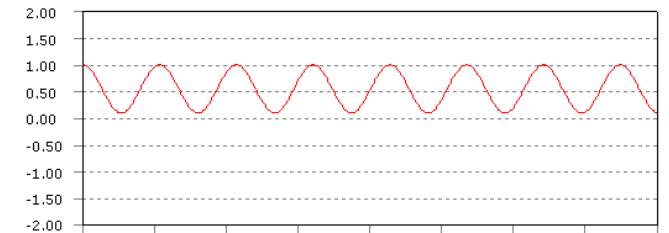
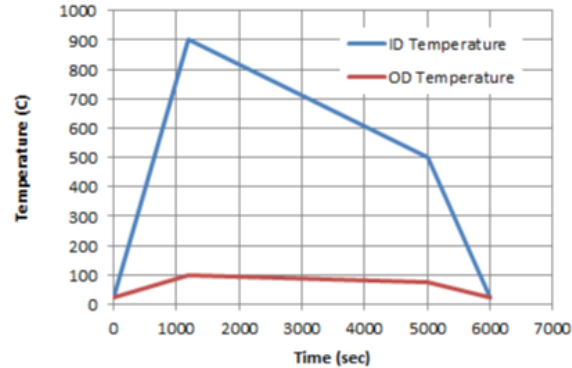
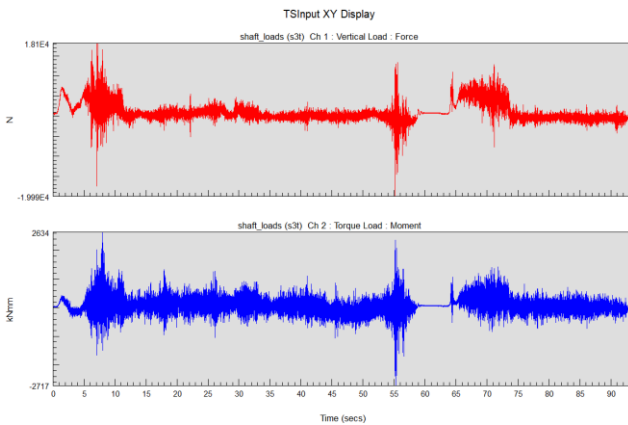
Graphs and Annotations:

- Temperature Graph (deg C):** Shows a ramp up from 22 to 215.2, followed by a gradual decrease. A red arrow points to the y-axis label "deg C".
- Thermal Stress Graph (MPa):** Shows a blue line fluctuating between approximately -66.88 and -3.431 MPa. A red box highlights the graph with the label "熱應力" (Thermal Stress).
- Pressure Measurement Graph (MPa):** Shows a green line with high-frequency oscillations between 0 and 478.3 MPa. A red box highlights the graph with the label "量測" (Measurement).
- Stress Fluctuation Graph (MPa):** Shows a red line with high-frequency oscillations between -65.84 and 466.4 MPa. A red box highlights the graph with the label "應力波動" (Stress Fluctuation).

Other Windows:

- Edit Load Map (LoadProviderTimeSeries1):** Shows "Edit Loading" set to "Advanced" and "Loading Type" set to "Time Series".
- TSinput XY Display:** Shows a zoomed-in view of the pressure history with a scale factor of 1. A red box highlights the graph with the label "應力波動".

- 潛變分析
- 高溫應力、應變疲勞分析
- 混合載荷類型：可以是溫度相關的
實測資料(TIME Series)、軟體設定的分析步階(TIME Step)、恆福加載(Constant)





謝謝聆聽!